OrCAD Layout® User's Guide

Product Version 10.5
July 2005

© 1985-2005 Cadence Design Systems, Inc. All rights reserved.
Printed in the United States of America.

Cadence Design Systems, Inc., 555 River Oaks Parkway, San Jose, CA 95134, USA

Trademarks: Trademarks and service marks of Cadence Design Systems, Inc. (Cadence) contained in this document are attributed to Cadence with the appropriate symbol. For queries regarding Cadence's trademarks, contact the corporate legal department at the address shown above or call 1-800-862-4522.

All other trademarks are the property of their respective holders.

Restricted Print Permission: This publication is protected by copyright and any unauthorized use of this publication may violate copyright, trademark, and other laws. Except as specified in this permission statement, this publication may not be copied, reproduced, modified, published, uploaded, posted, transmitted, or distributed in any way, without prior written permission from Cadence. This statement grants you permission to print one (1) hard copy of this publication subject to the following conditions:

1. The publication may be used solely for personal, informational, and noncommercial purposes;
2. The publication may not be modified in any way;
3. Any copy of the publication or portion thereof must include all original copyright, trademark, and other proprietary notices and this permission statement; and
4. Cadence reserves the right to revoke this authorization at any time, and any such use shall be discontinued immediately upon written notice from Cadence.

Disclaimer: Information in this publication is subject to change without notice and does not represent a commitment on the part of Cadence. The information contained herein is the proprietary and confidential information of Cadence or its licensors, and is supplied subject to, and may be used only by Cadence's customer in accordance with, a written agreement between Cadence and its customer. Except as may be explicitly set forth in such agreement, Cadence does not make, and expressly disclaims, any representations or warranties as to the completeness, accuracy or usefulness of the information contained in this document. Cadence does not warrant that use of such information will not infringe any third party rights, nor does Cadence assume any liability for damages or costs of any kind that may result from use of such information.

Restricted Rights: Use, duplication, or disclosure by the Government is subject to restrictions as set forth in FAR52.227-14 and DFAR252.227-7013 et seq. or its successor.

Contents

Welcome 21
How to use this guide 21
Symbols and conventions 22
Related documentation 23

Chapter 1 The Layout design flow 25

Layout master workflow 25
Board-level schematic 27
Component placement 27
Board routing 28
Post processing 29
Intertool communication 29

Chapter 2 Getting started 31

Opening a design 31
AutoECO 35
Starting a new board design with AutoECO 39
Using design libraries in AutoECO 40
Matching footprint pin names to schematic pin numbers 41
Resolving missing footprint errors 42
Resolving other AutoECO errors 43
Creating a design without a netlist 44
Saving a board 46
Closing a board and exiting Layout 47

Chapter 3 The Layout design environment 49

User interface 49
The Session Frame 49
The design window 50
The library manager 51
The session log 52

The toolbar 53
The status bar 57
Query window 57
Pop-up menu 58
Selecting and deselecting objects 59
The mouse, pointer and cursor 68
Density Graph window 70
Viewing the current coordinates 72
Viewing the place grid 72
Viewing the current layer 72
Using the postage stamp view 73
The Help System 73
Spreadsheets 74
Editing spreadsheet information 75
Statistics spreadsheet 76
Layers spreadsheet 77
Padstacks spreadsheet 78
Footprints spreadsheet 80
Packages spreadsheets 81
Components spreadsheet 82
Nets spreadsheet 83
Obstacles spreadsheet 84
Text spreadsheet 84
Error Markers spreadsheet 85
Drills spreadsheet 86
Apertures spreadsheet 87
Color spreadsheet 88
Post Process spreadsheet 89
Place Pass spreadsheet 91
Editing object properties 91
Setting environment preferences 92
Using and assigning colors 95
Assigning a color to an object 97
Assigning a color to a board layer 98
Changing the visibility of a layer or an object 99
Using color rules 102
Layer 0 (Zero) 104
Zooming 105
Numerical boundaries for Layout 107
Shortcuts 109
Global action shortcuts 109
File menu shortcuts 111
Edit menu shortcuts 111

View menu shortcuts 111
Tool menu shortcuts 112
Options menu shortcuts 114
Auto menu shortcuts 114
Window menu shortcuts 115
Window and spreadsheet shortcuts 115
Layer number shortcuts 116

Chapter 4 Layout files and file translation 119

System files 119
Design files 125
Library files 125
Report files 125
Netlist files 126
Board files 127
Board templates 127
Technology templates 127
Strategy files 132
MAX ASCII files 139
Exporting and importing ASCII files 139
MAX ASCII file general format 141
Attribute data format 143
Component data format 145
Net data format 148
Package and Symbol data format 151
Translating other file formats into Layout files 153
PADS Import 154
P-CAD Import 158
IDF Import 164
Protel Import 166
Protel 99SE Import 170
CadStar Import 173
Tango Import 177
SPECCTRA Import 181
GenCAD Import 182
PCB 386 Import 183
Translating Layout files into other file formats 185
IDF Export 186
SPECCTRA Export 187
GenCAD Export 188
GenCAM Export 189

IPC-356 Export 190
CadStar Export 191
ODB++ Export 194
PADS Export 195
P-CAD Export 198
Protel Export 201
Tango Export 204
HyperLynx Export 207
PCB 386+ Export 208
DXF import and export 209
Importing a DXF file 209
Layer Mapping for DXF translation 211
Adding text and obstacles with DXF import 214
Creating single-layer DXF files through the Post Processor . . 215
Creating multi-layer DXF files by exporting 216
Updating boards and libraries to Release 9 format 217
Converting pre-Layout v7.10 split planes 220

Chapter 5 Setting up the board 223

Creating a new Layout project from a Capture design 224
Using technology templates 225
Loading a technology template 226
Creating custom technology templates 226
Creating a board outline 229
Moving the datum 230
Setting units of measurement 231
Setting system grids 234
Adding mounting holes to a board 237
Defining the layer stack 238
Defining global spacing values 239
Defining padstacks 241
Defining vias 242
Defining an unused via 242
Assigning a via to a net 243
Changing via definitions 244
Placing a via 244
Viewing drill holes 245
Defining free vias 246
Setting net properties 247
Opening and editing nets 247
Enabling layers for routing 248

Setting net widths by layer 248
Setting reconnection order 249
Setting net spacing by layer 251

Chapter 6 Creating and editing obstacles 253

Creating obstacles 254
Selecting obstacles 255
Editing obstacles 256
Copying obstacles 256
Moving obstacles 257
Rotating obstacles 258
Mirroring obstacles 258
Exchanging the ends of obstacles 259
Moving segments 259
Creating circular obstacles 260
Deleting obstacles 261
Creating alignment targets 261

Chapter 7 Creating and editing text 263

Creating text 263
Moving text 268
Deleting text 268

Chapter 8 Design Reuse 269

Internal Design Reuse 270
Creating the Capture design for internal reuse 270
Annotating the design for internal reuse 271
Creating a netlist in Capture for Layout 273
Placing and routing the first instance of reuse 274
Applying internal design reuse to reuse targets 275
Internal design reuse example 277
Creating the Capture design for internal reuse 277
Annotating the design for internal reuse 281
Creating a netlist in Capture for Layout 287
Placing and routing the first instance of reuse 287
Applying internal design reuse to reuse targets 289

External Design Reuse 293
Creating the Capture design for external reuse 293
Annotating the design for external reuse 295
Creating a netlist and starting a new board 297
Applying external design reuse to reuse targets 298
External design reuse example 300
Creating the Capture design for external reuse 300
Annotating the design for external reuse 306
Creating a netlist and starting a new board 312
Applying external design reuse to reuse targets 312
Partial Design Reuse 320

Chapter 9 Placing and editing components 323

Preparing the board for component placement 323
Checking the board, place, and insertion outlines 324
Checking the place grid 325
Checking mirror layers and library layers 326
Weighting and color-coding nets 326
Checking gate and pin information 327
Securing preplaced components on the board 329
Creating height or group keepins and keepouts 330
Loading a placement strategy file 331
Grouping components for placement 332
Disabling the power and ground nets 333
Cross-placing components from Capture to Layout 333
Placing components manually 335
Placing components individually 335
Selecting the next components for placement 336
Building component clusters 337
Placing component groups 338
Minimizing connections to optimize placement 339
Copying, moving, and deleting components 339
Swapping components 340
Rotating components 340
Mirroring components 341
Placing components using a matrix 342
Editing components 344
Selecting an alternate footprint 347
Merging .MAX files into a design 348
Using autoplacement 349
Placing components using autoplacement 351

Using interactive placement commands 358
Adding footprints to the board 371
Checking placement 374

Chapter 10 Routing the board 377

Routing the board manually 378
Checking the board outline, via definitions, and routing and via grids 378
Loading and editing a routing strategy file 379
Changing board density using routing strategy files 381
Routing power and ground 381
Defining a DRC box 386
Fanout 388
Routing 394
Creating split planes 398
Verifying plane layer connections and disabling power and ground nets 401
Using manual routing tools 402
Using add/edit route mode 403
Using edit segment mode 404
Using interactive routing tools 406
Using shove track mode 407
Using auto path route mode 409
Creating duplicate connections 410
Manual routing techniques 411
Minimizing connections 411
Changing the colors of nets 412
Copying tracks 412
Removing tracks 412
Moving segments of tracks 414
Changing the widths of tracks 415
Forcing a net width on a layer 415
Drawing arcs 416
Adding vias 416
Adding a free via matrix 417
Changing vias 418
Changing free vias 418
Using tack points 420
Exchanging the ends of a connection 420
Locking routed tracks 421
Routing to an off-grid pad 421

Teeing into or out of an existing track 425
Making a ratsnest invisible 428
Routing hints 428
Creating and modifying nets 430
Creating nets 430
Splitting nets 431
Adding and deleting pins connected to nets 431
Disconnecting pins from nets 432
Generating test points 432
Checking routing 434
Using Route Spacing Violations 434
Viewing routing statistics 435
Resistor packages (pin swapping) 436
Autorouting 438

Chapter 11 Automatic routing using SPECCTRA 441

Overview 441
Prerequisites for automatic routing using SPECCTRA 442
Launching SPECCTRA from Layout 444
Performing fanout using SPECCTRA 447
Routing specific nets using SPECCTRA 448
Mitering wire corners using SPECCTRA 449
Autorouting the board using SPECCTRA 450
Running SPECCTRA using a customized .DO file 451
Generating SPECCTRA Reports 455

Chapter 12 Using thermal reliefs and copper pour zones . . . 457

Thermal reliefs 457
Defining thermal reliefs 458
Previewing thermal reliefs 459
Rules that apply to creating thermal reliefs 460
Forced thermal reliefs and preferred thermal reliefs 461
Using padstacks to create thermal reliefs 462
Copper pour zones 462
Designating a seed point 464
Creating a copper pour 465
Creating a circular copper pour 467
Specifying a hatch pattern 467
Refreshing a copper pour after editing the board 468

Using copper pour as a shield 469

Chapter 13 Ensuring manufacturability 473

Checking design rules 473
Investigating errors 474
Handling Known Errors 475
Delete Violating Tracks 477
Validating Gerber connectivity using an IPC-D-356 netlist 478
Cleaning up your design 482
Adding stackup data 483
Customizing stackup data in a template file 483
Add/modify stackup information on an existing board 484
Creating a new board with customized stackup settings 485
Modifying board stackup data 486
Deleting board stackup data 490

Chapter 14 Post processing 493

Renaming components 493
Annotating and cross probing 494
Back annotating 494
Forward annotating 495
Cross probing 498
Retaining non-electrical parts during AutoECO 500
Updating the footprint of a component using AutoECO 501
Comparing the connectivity of a Layout board file against the schematic netlist 502
User defined schematic attributes 502
Documenting board dimensions 505
Viewing the Post Process spreadsheet 507
Previewing layers 509
Previewing a layer 511
Viewing the drill drawing and drill chart 513
Restoring the design window view 514
Gerber plot preview 515
Moving the drill chart 516
Generating a drill tape 517
Using Run Post Processor 519
Creating reports 520
Printing and plotting 524

Chapter 15 Managing libraries and footprints 527

The library manager 529
Starting the library manager 530
Making libraries available for use 531
Viewing footprints 532
Searching Footprints 532
Creating a custom footprint library 535
Adding, copying, and deleting footprints 536
Creating and editing footprints 537
Setting a grid for the footprint pins 537
Creating a footprint 538
Editing footprints and footprint pins 556
Editing and copying padstacks 558
The Catalog Tool 561
Creating a catalog 562
Printing a catalog 564
Creating an Acrobat PDF file of an OrCAD Layout Library . . . 566
Creating design libraries 570

Chapter 16 Visual CADD 577

Manipulating the MECHANICAL mappings 577
Interactive DXF translation 578
Importing the DXF file 580
Translating components 581
Translating remaining elements 584
Stopping and resuming an incomplete translation 586
Importing a design from multiple DXF files 588

Chapter 17 GerbTool 591

The GerbTool Design File 591
Gerber to IPC-356 netlist comparison 592
Creating the Gerber files in Layout 593
Creating the IPC-356 netlist in Layout 594
Running the netlist comparison in GerbTool 594
Panelize 599
Manual panelization 601
Automatic panelization 601
Automatic venting 602

Virtual panelization 602
Teardrop Pads 603

Chapter 18 Dialog box descriptions 607

Add Color Rule dialog box 607
Add Component dialog box 608
Add Footprint dialog box 611
Add Free Via dialog box 613
Add Pad dialog box 615
Add Test Point dialog box 617
Advanced Options dialog box 619
Assign Via dialog box 619
Autodimension Options dialog box 620
Backup Settings dialog box 622
Check Design Rules dialog box 623
Circular Placement dialog box 626
Circular tab (Pad Array Generator) 630
Cleanup Design dialog box 632
Color dialog box 634
Comp Attachment dialog box 635
Component Attachment dialog box 636
Component Selection Criteria dialog box 637
Configure Design Library dialog box 639
Connector Stagger X tab (Pad Array Generator) 643
Connector Stagger Y tab (Pad Array Generator) 646
Copy Padstack Layer dialog box 648
Create Catalog dialog box 649
Create Catalog Additional Options dialog box 651
Create New Footprint dialog box 652
Create Stackup dialog box 652
Design Reuse dialog box 652
Drill Chart Properties dialog box 654
Drill Properties dialog box 655
Dual/Quad Inline tab (Pad Array Generator) 658
Edit Apertures dialog box 660
Edit Array Alphabet dialog box (Pad Array Generator) 662
Edit Component dialog box 663
Edit Footprint dialog box 665
Edit Free Via dialog box 668
Edit Layer dialog box 669
Edit Layer Strategy dialog box 675

Edit Net dialog box 678
Edit Obstacle dialog box 684
Edit Pad dialog box 687
Edit Padstack dialog box 690
Edit Padstack Layer dialog box 695
Edit Place Pass dialog box 699
Edit Route Pass dialog box 703
Edit Spacing dialog box 709
Edit Test Point dialog box 711
Fanout Settings dialog box 712
Fanout tab (SPECCTRA Automatic Router Parameters) 715
Find and Select Item dialog box 719
Find Coordinate or Reference Designator dialog box 720
Footprint Selection dialog box 720
Free Via Matrix Settings dialog box 721
Free Via Selection Criteria dialog box 723
GenCAD to Layout dialog box 724
General tab (SPECCTRA Automatic Router Parameters) 725
Generate Reports dialog box 727
Gerber Preferences dialog box 731
Grid Array tab (Pad Array Generator) 733
Group Settings dialog box 736
IDF to Layout dialog box 738
Hatch Pattern dialog box 740
Jumper Lengths dialog box 741
Layers Enabled for Routing dialog box 742
Layout MAX to HYP window 743
Layout to GenCAD dialog box 744
Layout to GenCAM dialog box 745
Layout to IPC-356 dialog box 746
Layout to IDF dialog box 747
Layout to ODB++ dialog box 749
Layout to SPECCTRA dialog box 751
Library Conversion Warning dialog box 752
Link Footprint to Component dialog box 752
Manual Route Strategy dialog box 753
Modify Connections dialog box 755
Modify Nets dialog box 756
Net Selection Criteria dialog box 756
Net Spacing by Layer dialog box 757
Net Widths by Layer dialog box 758
Obstacle Selection Criteria dialog box 759
Package Edit dialog box 760

Pad Array Generator dialog box 764
Pin Attachment dialog box 766
Place Settings dialog box 766
Plating Properties dialog box 768
Post Process Settings dialog box 769
Print Catalog dialog box 773
Print/Plot dialog box 774
Properties (Layer) dialog box 778
Properties (Solder Paste) dialog box 779
Properties (Solder Mask) dialog box 780
QFP/Chip Carrier tab (Pad Array Generator) 780
Reconnection Type dialog box 783
Rename Direction dialog box 785
Replace dialog box 786
Replace Footprint dialog box 786
Replace Footprint (footprint name) dialog box 789
Route Settings dialog box 789
SeedVia tab (Preferences) 795
Miter Corners tab (Preferences) 795
Route Strategy tab (SPECCTRA Automatic Router Parameters) . 796
Router Setup tab (SPECCTRA Automatic Router Parameters) . . 811
Save Footprint As dialog box 812
Save padstack - Select library dialog box 813
Search Footprint dialog box 813
Select Footprint dialog box 816
Select Footprint dialog box (for Design Library) 817
Select Layer dialog box 818
Select Next dialog box 818
Select Padstacks dialog box 819
Select Padstack dialog box (Pad Array Generator) 820
SPECCTRA Reports dialog box 822
SPECCTRA to Layout dialog box 825
Stackup Editor dialog box 826
Sweep Edit dialog box 829
System Settings dialog box 834
Test Point Settings dialog box 835
Test Point Selection Criteria dialog box 836
Text Edit dialog box 837
Text Selection Criteria dialog box 839
Thermal Relief Settings dialog box 840
Track Width dialog box 841
User Preferences dialog box 842
Via Selection dialog box 847

Workspace Settings dialog box 847

Chapter 19 Layout Session Window Commands 849

File Menu Commands (LSession) 849
Import command (File menu) 849
Export command (File menu) 856
1, 2, 3, 4 file commands (File menu) 863
View Menu Commands (LSession) 864
Toolbar command (View menu) 864
Status Bar command (View menu) 864
Tools Menu Commands (LSession) 865
Library Manager command (LSession: Tools menu) 865
Create (Catalog) command 866
Print (Catalog) command 866
Define Stackup command 866
Configure Design Library command 867
OrCAD Capture command (Tools menu) 867
IntelliCAD command 867
SmartRoute command (Tools menu) 868
GerbTool command (Tools menu) 868
ECOs command (Tools menu) 869
Edit App Settings command (Tools menu) 880
Reload App Settings command (Tools menu) 881

Chapter 20 Layout Commands 883

File Menu Commands 883
New command (File menu) 884
Open command (File menu) 885
Load command (File menu) 885
Save command (File menu) 889
Save As command (File menu) 889
Backup command (File menu) 892
Close command (File menu) 893
Print/Plot command (File menu) 893
Library Manager command (File menu) 894
Text Editor command (File menu) 895
Exit command (File menu) 895
Edit Menu Commands 895
Undo command (Edit menu) 896

Copy command 896
Paste command 897
Delete command 897
Find/Goto command (Edit menu) 897
Select Any command (Edit menu) 898
Select Next command (Edit menu) 899
Clear Selections command (Edit menu) 900
End command (Edit menu) 900
Properties command (Edit menu) 901
View Menu Commands 901
Design command (View menu) 902
Density Graph command 902
Preview command (View menu) 904
High Contrast command (View menu) 904
Clear Screen command (View menu) 904
Redraw command (View menu) 905
Query Window command (View menu) 905
Database Spreadsheets command (View menu) 906
Zoom All (Fit) command (View menu) 906
Zoom Center command (View menu) 907
Zoom In command (View menu) 907
Zoom Out command (View menu) 908
Zoom Previous command (View menu) 908
Zoom DRC/Route Box command (View menu) 909
Visible <> Invisible command (View menu) 909
Tool Menu Commands 909
Layer Flyout Commands 910
Block command 911
Cluster Flyout Commands 916
Group Flyout commands 917
Matrix Flyout Commands 921
Component Flyout Commands 923
Gate Flyout Commands 930
Footprint Flyout Commands 930
Padstack Flyout Commands 932
Pin Flyout Commands 935
Aperture Flyout Commands 938
Net Flyout Commands 939
Connection Flyout Commands 944
Track Flyout Commands 947
Track Segment Flyout Commands 950
Jumper Flyout Commands 952
Via Flyout Commands 954

Test Point Flyout Commands 958
Drill Chart Flyout Commands 960
Text Flyout Commands 960
Dimension Flyout Commands 964
Measurement Flyout Commands 966
Obstacle Flyout Commands 967
Error Flyout Commands 972
Options Menu Commands 973
System Settings command 974
Colors command 974
Color Rules command 975
Auto Backup command 975
Stackup Settings command 976
Global Spacing command 976
Placement Strategy command 976
Place Settings command 977
Route Strategies Flyout Commands 977
Route Settings command 980
Fanout Settings command 981
Thermal Relief Settings command 981
Jumper Settings command 981
Free Via Matrix Settings command 982
Test Point Settings command 982
Components Renaming command 982
Gerber Settings command 983
Post Process Settings command 983
User Preferences command 983
Auto Menu Commands 984
Refresh Flyout Commands 984
Place Flyout Commands 987
Unplace Flyout Commands 990
Fanout Flyout Commands 991
Autoroute Flyout Commands 993
Autoroute SPECCTRA Flyout Commands 996
Unroute Flyout Commands 998
Design Rule Check command 1000
Delete Violating Tracks Flyout Commands 1000
Cleanup Design command 1001
Rename Components command 1002
Back Annotate command 1002
Run Post Processor command 1002
Create Reports command 1003
Window Menu Commands 1003

Cascade command (Window menu) 1004
Tile command (Window menu) 1004
Arrange Icons command (Window menu) 1004
Half Screen command (Window menu) 1004
Reset All command (Window menu) 1005
1, 2, 3, 4 commands (Window menu) 1005
Help Menu Commands 1005
Layout Help command 1006
What’s New command 1006
Known Problems and Solutions command 1006
Web Resources Commands 1006
Learning Layout command 1007
Manuals command 1007
About Layout command 1007
View Spreadsheet Toolbar Button Commands 1008
Strategy command (Spreadsheets) 1008
Pop-up menus 1010
Block tool pop-up commands 1011
Component tool pop-up menu commands 1012
Gate tool pop-up menu commands 1013
Pin tool pop-up menu commands 1014
Obstacle tool pop-up menu commands 1014
Text tool pop-up menu commands 1015
Connection tool pop-up menu commands 1015
Error tool pop-up menu commands 1015
Track tool pop-up menu commands 1016
Track Segment tool pop-up menu commands 1017
End Command (pop-ups) 1018

Before you begin

Welcome

OrCAD Layout is a powerful printed circuit board layout tool that is a part of a full line of design and simulation tools available from OrCAD. OrCAD Layout makes it easy to place, route and prepare printed circuit boards for fabrication.

How to use this guide

This guide is designed to make the most of the advantages of online books. The table of contents, index, and cross references provide instant links to the information you need. Just click on the text and jump.

If you find printed paper helpful, print only the section you need at the time. When you want an in-depth tutorial, print the example. When you want a quick reminder of a procedure, print the procedure.

Symbols and conventions

OrCAD printed documentation uses a few special symbols and conventions

Notation	Examples	Description
CTRL+R	Press CTRL+R	Means to hold down the CTRL key while pressing R.
ALT, F, O	From the File menu, choose Open (ALT, F, O)	Means that you have two options. You can use the mouse to choose the Open command from the File menu, or you can press each of the keys in parentheses in order: first ALT, then F, then O.
Monospace font	In the Part Name text box, type PARAM.	Text that you type is shown in monospace font. In the example, you type the characters P, A, R, A, and M.
UPPERCASE	In Capture, open CLIPPERA.DSN.	Path and filenames are shown in uppercase. In the example, you open the design file named CLIPPERA.DSN.
Italics	In Capture, save *design_name*.DSN.	Information that you are to provide is shown in italics. In the example, you save the design with a name of your choice, but it must have an extension of .DSN.

Related documentation

In addition to this guide, you can find technical product information in the online Help, the online interactive tutorial, online books, OrCAD's technical web site, as well as other books. The table below describes the types of technical documentation provided with Layout.

This documentation component . . .	Provides this . . .
This guide— OrCAD Layout User's Guide	A comprehensive guide for understanding and using the features available in Layout.
Online Help	Comprehensive information for understanding and using the features available in Layout. You can access Help from the Help menu in Layout, by choosing the Help button in a dialog box, or by pressing F1. Topics include: ■ Explanations and instructions for common tasks. ■ Descriptions of menu commands, dialog boxes, tools on the toolbar and tool palettes, and the status bar. ■ Reference information.
Online interactive tutorial	A series of self-paced interactive lessons. You can practice what you've learned by going through the tutorial's specially designed exercises that interact directly with Layout. You can start the tutorial by choosing Learning Layout from the Help menu.
Online *OrCAD Layout Quick Reference*	Concise descriptions of the commands, shortcuts, and tools available in Layout.
Allegro PCB Router User Guide	This guide contains high-level procedural and conceptual information on SPECCTRA, including introductory information regarding user interface usage, file management topics and work flow diagrams.

This documentation component . . .	Provides this . . .
Allegro PCB Router Command Reference	This reference is a comprehensive source of information for both menu and console commands in the Place and Route environments in SPECCTRA. Console commands are fully described using syntax diagrams and command examples.

The Layout design flow

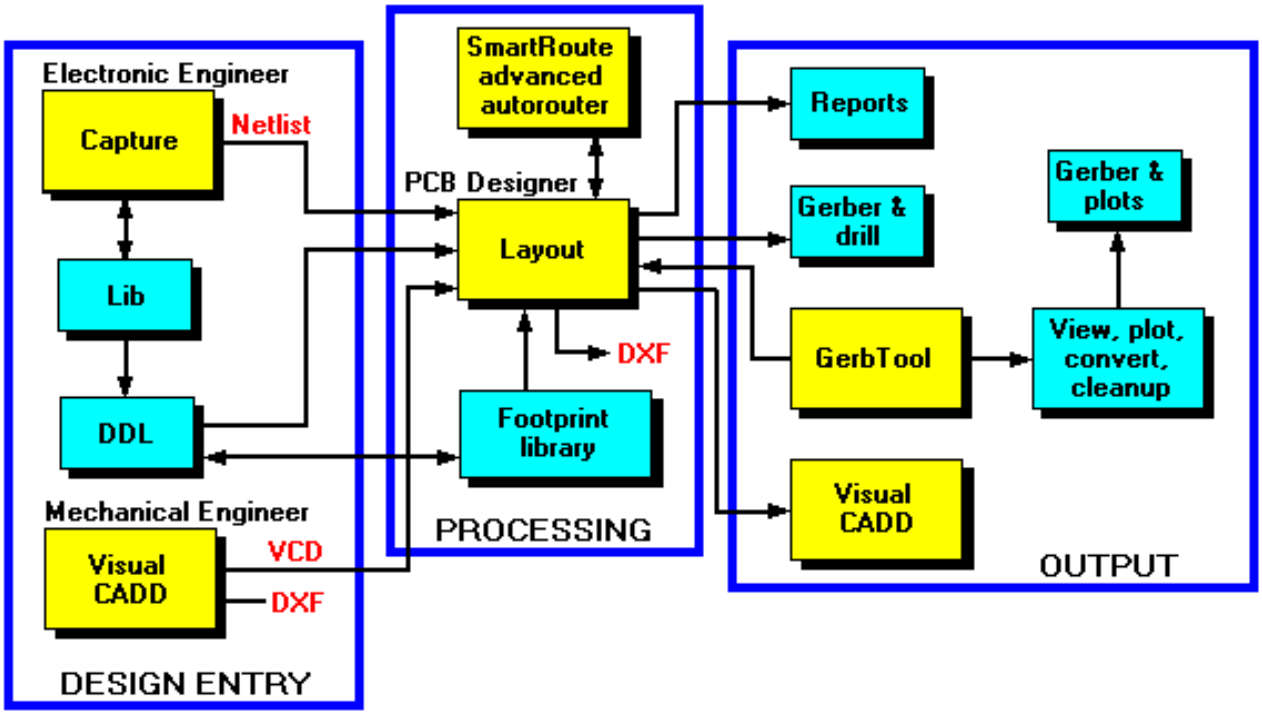
1

Layout master workflow

Layout supports every phase of the design process. A typical printed circuit board design flow has five key phases:

- Board-level schematic
- Component placement
- Board routing
- Post processing

- Intertool communication



Related topics

Getting started

The Layout design environment

Layout files and file translation

Setting up the board

Creating and editing obstacles

Creating and editing text

Placing and editing components

Routing the board

Using thermal reliefs and copper pour zones

Ensuring manufacturability

Post processing

Managing libraries and footprints

Visual CADD

GerbTool

Board-level schematic

Using a schematic capture tool, such as OrCAD Capture, you can create a Layout-compatible netlist that includes preset design rules to guide logical placement and routing. This gives you the ability to specify critical design rules at the schematic level, such as component locations, net spacing criteria, component group information, net widths, and routing layers, and bring them into Layout in a netlist. If the schematic netlist changes, you can reload it. Layout's AutoECO (automatic engineering change order) utility updates the board without harming finished work.

Related topics

Component placement

Board routing

Post processing

Intertool communication

Component placement

Whether you choose to use Layout's manual placement tools, or the interactive and autoplacement utilities (available in Layout Plus only), you have ultimate control of the component placement process. You can place components individually or in groups.

During autoplacement, Layout's *shove* capability moves components out of your way automatically while adhering to

design rule check (DRC) guidelines. You can autoplace components individually, by area, or you can autoplace the entire board.

Related topics

Board-level schematic

Board routing

Post processing

Intertool communication

Board routing

With Layout, you can route your board manually, or you can use Layout's interactive and automatic routing tools (available in Layout Plus and Layout only).

Using manual routing, you guide the routing process and manually route each track. Then you optimize routing using a variety of manual routing commands.

In interactive routing, you still control the routing of individual tracks, but can take advantage of Layout's automatic routing technologies, such as *push-and-shove*, which moves tracks to make space for the track you are currently routing.

If you choose to use Layout's autorouter, you can interrupt routing at any time to manage and control the routing process. You can autoroute a single track, a selected area of the board, a group of nets, or the entire board.

Related topics

Board-level schematic

Component placement

Post processing

Intertool communication

Post processing

In Layout, all of your output settings are stored in a spreadsheet that you can call up and revise. You can give layer-by-layer instructions for writing to Gerber files, DXF files, or hardcopy devices.

Layout produces more than twenty standard reports, including fabrication drawings, assembly drawings, and pick-and-place reports. In addition, you can create custom reports of your own.

Related topics

Board-level schematic

Component placement

Board routing

Intertool communication

Intertool communication

Layout has the ability to communicate interactively with OrCAD Capture using intertool communication (ITC).

You can use intertool communication to communicate updated schematic information to Layout at any stage of the design process. Also, you can back annotate board data to Capture from Layout.

Intertool communication supports *cross-probing* to facilitate design analysis. If you select a signal or part in Capture, the corresponding signal or part is highlighted in Layout, and vice versa.

Related topics

Board-level schematic

Component placement

Board routing

Intertool communication

Post processing

Getting started

2

This chapter describes how to:

- load a board template
- load a netlist
- open a board
- save a board
- close a board
- exit Layout

Related topics

The Layout design flow

The Layout design environment

Opening a design

You can open a new design or an existing design. When you open a new board design, Layout prompts you to choose a *template* and a schematic netlist. A board template provides the framework within which you can create a board design. A netlist describes the parts and interconnections of a schematic design.

A board template (*file_name*.TPL) contains a board outline and design rules from Layout's default *technology template*, DEFAULT.TCH. DEFAULT.TCH, described in *Technology templates,* contains the following parameters, among others:

- 62-mil pads
- 12-mil tracks
- 12-mil spacing

The board templates, located in the LAYOUT/DATA directory, offer numerous, unique board outlines. The board outline titles correspond to the filenames of the board templates that contain them.

Note: If you cannot use any of the board outlines provided with Layout, you can create your own board outline. In this case, load a technology template (.TCH) instead of a board template (.TPL) when you open the new design. Then, create your own board outline by following the instructions in Creating a board outline on page 229.

If you choose to load one of the board templates (board outlines) provided with Layout, but DEFAULT.TCH is not suitable for your type of board, you can load a technology template to match the characteristics of your board, including manufacturing complexity and component type. You can load a technology template after you open the board.

Note: For more information on netlist files, board files, and technology templates, and for a complete list of the technology templates provided with Layout, see Design files.

Note: If you load a technology template after loading a board template, you can save the result as a custom technology template for use with future designs. See Using technology templates on page 225 for more information.

A netlist file describes the interconnections of a schematic design using the names of the signals, components, and pins. A netlist file (.MNL) contains the following information:

- Footprint names
- Electrical packaging
- Component names
- Net names
- The component pin for each net
- Net, pin, and component properties

You can create a Layout netlist directly in Capture, or you can import Layout-supported netlists using a translator that corresponds to your schematic program. The translator creates the file *design_name*.MNL.

The AutoECO (Automatic Engineering Change Order) process combines a board template (.TPL) and a schematic netlist (.MNL) to produce a Layout board file (.MAX) that contains all of the board's physical and electrical information.

Note: Running multiple copies of Layout is not recommended. When multiple copies of Layout are running, data is shared between the two copies. Performing processes like AutoECO or making edits in the Library Manager can cause changes in other open copies.

Figure 2-1 illustrates the process for opening a new design.

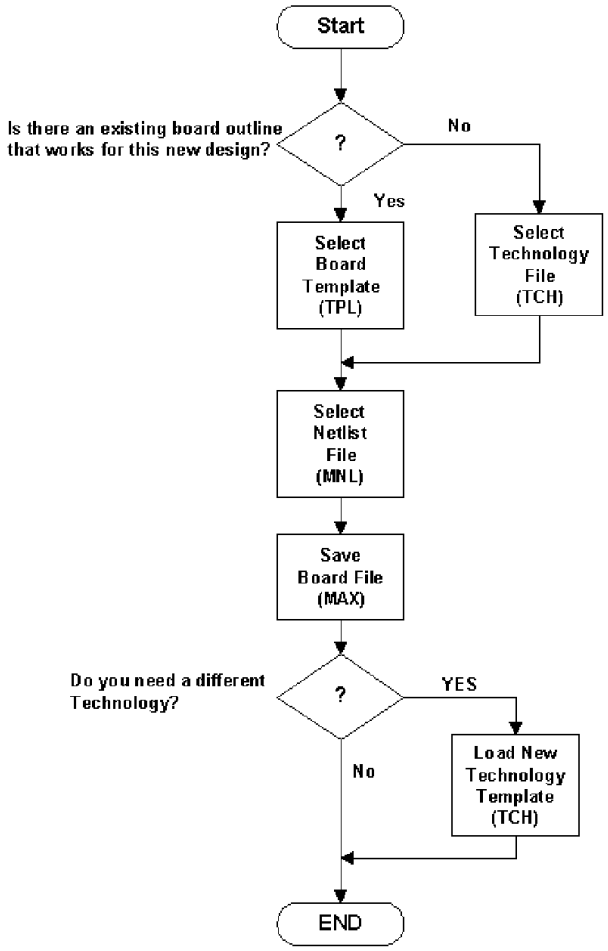


Figure 2-1 Opening a new board design in Layout.

To open a new design

1 From the File menu, choose New. The Load Template File dialog box appears.

Note: If you do not want to load one of the board outlines provided with Layout, load a technology template instead (.TCH). For more information about technology templates, and for a complete list of the technology templates provided with Layout, see *Technology templates*.

2 Select a board template (.TPL or .TCH), then choose the Open button. The Load Netlist Source dialog box appears.

3 Select a netlist file (.MNL), then choose the Open button. The Save File As dialog box appears.

4 Supply a name for the new board file (.MAX), then choose the Save button. AutoECO runs automatically, and displays its progress in an ASCII report file (.LIS). If there are no AutoECO errors, the new board opens in Layout's design window.

Related topics

Creating a design without a netlist

AutoECO

AutoECO enables you to forward annotate information from a design in Capture to a board in Layout. AutoECO also resolves "pin-to-pin" conflicts that may arise, due to pins that are missing from a chosen footprint, or pins that are named differently in Capture than they are in Layout (for example, a diode may have pins named pin A and pin C in Capture, but named pin 1 and pin 2 in Layout).

AutoECO selectively communicates information from Capture to Layout, or from one printed circuit board to another. In other words, when bringing data into Layout from Capture, you can choose the appropriate AutoECO option to annotate only the data that has changed. This way, you can avoid inadvertently overriding board data that you do not want to change. You can also choose from two AutoECO options that communicate information between printed circuit boards.

You can use AutoECO to start a new board design or forward annotate schematic design changes. These tasks are completed with subtle variations through each of the seven AutoECO options.

The AutoECO options are listed below. The first six options transfer data from Capture to Layout during forward

annotation. The last option, AutoECO/Net Attrs, communicates information between printed circuit boards.

- AutoECO - adds and deletes components and nets, but does not override board attributes. This version is used when Layout runs AutoECO automatically.
- AutoECO/Override Attrs - creates a new board or merges new components and connections with an existing board. It overrides all of the attributes in an existing board. Components and nets that are no longer needed are deleted.
- AutoECO/Override Coords - creates a new board or merges new components and connections with an existing board. It overrides all of the placement coordinates in an existing board, and reconciles all components and nets with the netlist.

  Note: The COMPSIDE user attribute is not updated with the AutoECO/Override Coords ECO. Use the AutoECO/Override Attrs command to update Schematic attributes.
- AutoECO/Override All - creates a new board or merges new components and connections with an existing board. This ECO reconciles all components, nets and properties with the netlist. Also moves all components to the netlist coordinates. Components and nets that are no longer needed are deleted.
- AutoECO/Add Only - adds components and nets, but does not override the board attributes. Properties on existing components are not updated. This option is useful when the board is near completion and you wish to add some parts (for example, a bypass capacitor).
- AutoECO/Add Override - adds new components and new nets, and overrides all component and net properties. Unused components and nets are not deleted.
- AutoECO/Net Attrs - transfers all net properties such as width, weight, spacing-per-layer, width-per-layer, and reconnection type from one board to another. Components and net names are not changed.

The following tables summarize the capabilities of each type of AutoECO that communicates data from Capture to Layout (.MNL to .MAX) during forward annotation, and between printed circuit boards during translation.

	AutoECO MNL to MAX	Override Attrs MNL to MAX	Override Coord MNL to MAX	Override All MNL to MAX
Components	Match schematic	Match schematic	Match schematic	Match schematic
Nets	Match schematic	Match schematic	Match schematic	Match schematic
Obstacles	No changes	No changes	No changes	No changes
Text	No changes	No changes	No changes	No changes
Placement	No changes except new	No changes	Match schematic	Match schematic
Component Properties except Coordinates	No changes except new	Match schematic	No changes except new	Match schematic
Component Coordinates	No changes except new	No changes except new	Match schematic	Match schematic
Net Properties	No changes except new	Match schematic	No changes except new	Match schematic

	Add Only MNL to MAX	Add Override MNL to MAX	Net Attrs MNL to MAX
Components	Match schematic except deletes	Match schematic except deletes	No changes
Nets	Match schematic except deletes	Match schematic except deletes	No changes
Obstacles	No changes	No changes	No changes
Text	No changes	No changes	No changes

	Add Only MNL to MAX	Add Override MNL to MAX	Net Attrs MNL to MAX
Placement	No changes except new	Match schematic except deletes	No changes
Component Properties except Coordinates	No changes except new	Match schematic except deletes	No changes
Component Coordinates	No changes except new	Match schematic except deletes	No changes
Net Properties	No changes except new	Match schematic except deletes	Match schematic

Match schematic

All existing data on the PCB in this category is made to match the schematic page, including additions and deletions.

Match schematic except deletes

All existing data on the PCB in this category is made to match the schematic page, including additions, but not deletions.

No changes except new

All existing data on the PCB in this category remains the same, except additions are made from the schematic page.

No changes

All data in this category of the PCB remains the same.

Add/replace from new board

Any matches to existing data on the old PCB are replaced from new PCB design, including additions, but not deletions.

Match original board where possible

Any matches to existing data on the old PCB are replaced from new PCB design, but no additions or deletions are made.

Note: You may substitute another MAX file where the AutoECO program calls for an .MNL file. The components, netlist, and properties will pass forward from the MAX file exactly as if you had started with an .MNL file.

Layout merges the files based on the type of AutoECO you have chosen.

About duplicate pin errors

If in previous versions of Layout power pins were marked as "passive" to show visibility. This can cause duplicate pin errors during AutoECO. Capture now has visibility control for pins, so you should remove all passive markers on power pins and set the visibility to the same setting for all the power pins in a package.

Starting a new board design with AutoECO

When AutoECO is used to create a new design, the process combines a board template (.TPL) and a schematic netlist (.MNL) to produce a Layout board file (.MAX). The template contains the board's physical characteristics and the netlist the electrical.

To start a new board with AutoECO, the following must be completed:

- Create the schematic design using Capture, and resolve all DRC warnings and errors.
- Create a .MNL netlist and resolve all netlist warnings and errors.

Note: Unresolved warnings and errors can cause AutoECO to fail. If necessary, create a netlist in wirelist format to check for errors. The wirelist format is a readable ASCII file.

To use AutoECO to start a new board design

1. In the LSession File menu, choose New. The AutoECO dialog box appears.
2. Enter the name and path to the input template file (.TPL), technology file (.TCH) or board file (.MAX), or click Browse to select the file.
3. Browse to the Capture project folder and select the Layout netlist (.MNL) file.
4. Enter a name for the new board file (.MAX) you are creating.
5. Click the Apply ECO button.

If there are no problems, a new .MAX file is created. You can place components and route the design. However, if there are discrepancies, the .MAX file is not created and two reports are created:

- filename.LIS – a list of the processing that occurred and the discrepancies found.
- filename.ERR – a list of the discrepancies found.

These files are in the same directory as your target .MAX file. The errors must be resolved. A .MAX file will not be created until all problems are resolved.

Using design libraries in AutoECO

If you use a design library, you can limit the AutoECO process to only the design library. This saves time in the design cycle as the AutoECO process needs to scan only the design library instead of scanning all the libraries made available for use in Layout.

For more information on creating design libraries, see Creating design libraries on page 570.

To use design libraries in AutoECO

1. Select the Use design library only check box in the AutoECO dialog box.
2. Enter the path and filename of the design library (.LLB) file for the design or click the Browse button to select the file.

   Note: If you want to edit the design library, click the Edit Library button to display the Configure Design Library dialog box. Make the required changes to the design library and click the Save button to save the changes.
3. Click the Apply ECO button to run AutoECO.

   The Configure Design Library dialog box appears if any Capture part is not mapped to a footprint in the design library.

   Ensure that footprints are selected for all parts, then click the Save button to save the design library and continue the AutoECO process.

   Note: You can only search and replace footprints for parts for which no footprint was selected earlier. You cannot replace the footprints you selected for parts when you created or edited the design library.

Related topics

Creating design libraries

Matching footprint pin names to schematic pin numbers

Pin names in the schematic must match the footprint pin numbers in the footprint library files. For example, a diode in the schematic might have pins called Anode and Cathode, while the actual footprint has corresponding pin names of Ano and Cath, or 1 and 2. These differences must be reconciled or the design will not load. To correct this situation, either:

- change the symbol pin names in the schematic to match the footprint pin names in the Layout library.

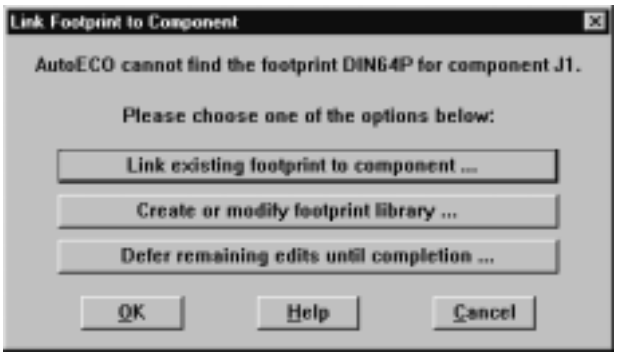
- change the footprint pin names in the library to match the symbol pin names.

Each device in the schematic describes an electrical part. For example, a description could be 74LS00. Electrical parts are matched to footprints in one of three ways:

- The part contains a footprint attribute, such as DIP14, that matches a footprint found in the Layout footprint library.
- The part name 74LS00 is linked to a footprint in the SYSTEM.PRT file located in the LAYOUT\DATA subdirectory.
- If you are in the process of running AutoECO, and Layout is unable to find a designated footprint, the Link Footprint to Component dialog box appears.

Resolving missing footprint errors

If you are in the process of running AutoECO and it is unable to find a designated footprint, the Link Footprint to Component dialog box appears. Choose one of the options in the dialog box (described below) to resolve the error, so that the AutoECO process can continue.


Link Footprint to Component
AutoECO cannot find the footprint DIN64P for component J1.
Please choose one of the options below:
Link existing footprint to component ...
Create or modify footprint library ...
Defer remaining edits until completion ...
OK
Help
Cancel

Link existing footprint to component

Displays the Select Footprint dialog box, within which you can locate and select the desired footprint, then choose the OK button to return to AutoECO. (Choose the Add button in the

Select Footprint dialog box to add additional footprint libraries, if necessary.)

Create or modify footprint library

Opens the library manager, which you can use to create or modify footprint libraries. When you're finished, exit the library manager (from the File menu, choose Exit) to return to AutoECO.

Defer remaining edits until completion

Continues to run AutoECO, then checks for errors at its completion. Layout reports missing footprints in an ASCII file (*design_name*.ERR).

Resolving other AutoECO errors

There are two other problems that can occur during the AutoECO process when opening a design.

- Mounting holes disappear from the board when you run AutoECO.
- The pin numbers from the schematic do not match the pad names in Layout.

If an object, such as a mounting hole, is on the board but not in the schematic, specify it as Not in Netlist in the Edit Component dialog box. Otherwise, it may be deleted when you run AutoECO.

To define a component as Not in Netlist

1. Choose the spreadsheet toolbar button, then choose Components.
2. Locate and double-click on the component in the spreadsheet. The Edit Component dialog box appears.
3. In the Component Flags group box, check the Not in Netlist option, then choose the OK button.

Note: When AutoECO finds errors, it creates and displays an .ERR file. To correct pin problems, you can return to Capture to change numbering, then repeat the forward annotation procedure. Or, you can edit the footprint in Layout's footprint library, then recreate the board file. If you encounter footprint errors, ensure that the footprint name in Capture matches the footprint name in Layout.

Pin numbers in the schematic must match the footprint pin names in the footprint library files. For example, a diode in the schematic might have pins named Anode and Cathode, while the actual footprint has corresponding pin names of Ano and Cath. These differences must be reconciled or the design will not load. To correct this situation, do one of two things.

- Change the symbol pin names in the schematic to match the footprint pin names in the Layout library.
- Change the footprint pin names in the library to match the symbol pin names.

To open an existing board

1. From the File menu, choose Open. The Open Board dialog box appears.
2. Locate and select an existing board (.MAX), then choose the Open button.
3. If necessary, respond to the message asking if you want to update the board because the netlist has changed. The board opens in the design window.

Creating a design without a netlist

To create a board layout without a netlist, you must open a new board, load a technology file, place the components, and create nets. Then you can export the board file and import it as a .MAX file.

To open a new board file

1. In the Layout shell, choose New Design from the Tools menu.

   The Load Technology File dialog box displays.

2. Select a technology file.

   The Load Netlist source dialog box displays.

3. Choose the Cancel button.

   A new board opens in the design window.

To place components on the board

1. Choose the Component tool.
2. Choose New from the pop-up menu.

   The Add Component dialog box displays.

3. Choose the Footprint button.

   The Select Footprint dialog box displays.

4. Select a footprint from the Footprint drop-down list.
5. Choose the OK button twice to dismiss the dialog boxes.

   The part is attached to the cursor.

6. Position the part and place it on the board by clicking the left mouse button.
7. Repeat for each part you want to place.

To create nets using the Nets spreadsheet

1. Choose the Spreadsheets toolbar button.
2. Select Nets from the drop-down list.

   The Nets spreadsheet appears.

3. Select the DEFAULT net and choose New from the pop-up menu.

   A net is added.

4 Select the net and choose Properties from the pop-up menu.

The Edit Net dialog box displays.

5 In the Net Name group box, enter a new name for the net and choose the OK button.

6 Repeat the process for each net you want to create.

7 Choose the Connections toolbar button.

8 Make the connections manually, from pin to pin, using the net names from the Nets spreadsheet.

9 Repeat the process for each connection you want to create.

Saving a board

To save a new board

1 From the File menu, choose Save As. The Save File As dialog box appears.

2 Select a folder, enter a filename in the File name text box, then choose the Save button. The board is saved, and remains open in the design window.

To save an existing board

1 From the File menu, choose Save. The board is saved in the directory it was opened from, and remains open in the design window.

To save a copy of a board

1 From the File menu, choose Save As. The Save File As dialog box appears.

2 Select a folder, enter a filename in the File name text box, then choose the Save button. A copy of the board is created. The copy of the board appears in the design window and the original file is closed.

Closing a board and exiting Layout

To close a board

1. From the File menu, choose Close. Layout asks if you want to save your changes.
2. Choose either the Yes or No button. Layout displays an empty board in the design window.

To exit Layout

1. From the File menu, choose Exit. Layout asks if you want to save your changes.
2. Choose either the Yes or No button. Layout quits.

The Layout design environment

3

This chapter describes the things you should know to find your way around in Layout. It describes the design window, the library manager, the spreadsheets, and other items. It also introduces you to the toolbar, and to general Layout concepts such as selecting and editing objects, and using pop-up menus.

User interface

The Session Frame

Using the session frame, you can enter the Layout design window or the library manager or footprint editor, and you can access the utilities for design translation or for AutoECO.

To open an existing design, choose the Open command on the File menu, then navigate to the directory that holds your design file.

In addition, you can easily edit your LSESSION.INI file. This file must point to the executables that you can access from the session frame. If you change your default setup, you need to also edit your LSESSION.INI to match the change. To edit the file, choose Edit App Settings on the Tools menu, make your changes in Notepad, and save the file. Then you need simply

choose Reload App Settings to configure Layout for future work sessions.

Related topics

Post processing

Visual CADD

GerbTool

The design window

The design window provides a graphical display of the printed circuit board, and is the primary window you use when designing your board. It also provides tools to facilitate the design process, such as the tools to update components or check for design rule violations. The design window appears when you open a new or existing board.

In the design window, you can use the various tools (from the Tool Menu) to implement layout, routing, and other board design tasks.

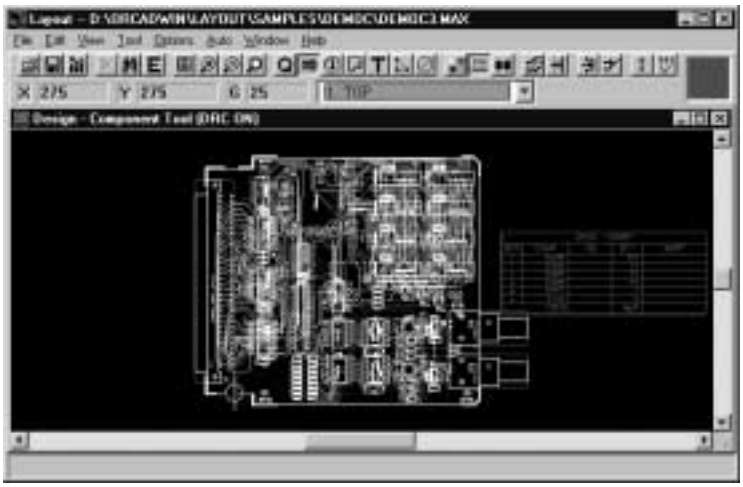
The X and Y grid coordinates for the pointer are displayed at the right edge of the window just below the toolbar.

The grid spacing value is displayed to the right of the X and Y grid coordinates. This value reflects the units of measurement you specify in the Display Units dialog box.

The layer drop-down list is to the right of the grid spacing value. You can view another layer by choosing from the list.

The status bar in the lower left corner of the window displays the grid coordinates for the pointer and memory use. It also displays the name and type of any selected component,

obstacle, pin, track, or text plus the distance from the object's original position.



Related topics

Database Spreadsheets command (View menu)

Strategy command (Spreadsheets)

Density Graph command

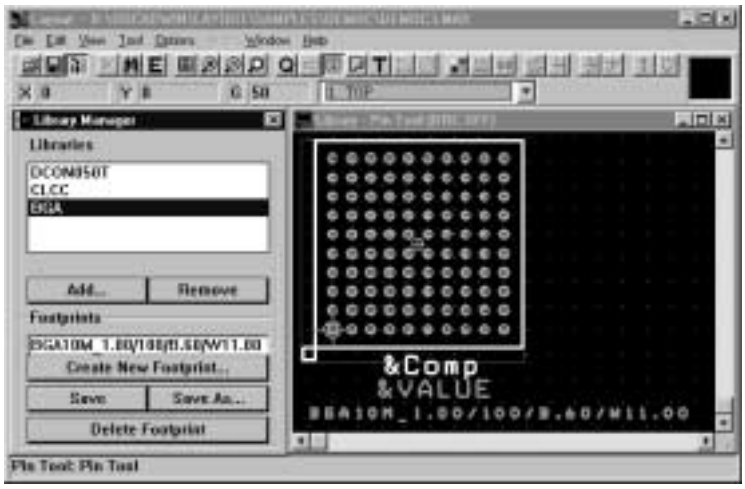
The library manager

The library manager is used to view, create, and edit footprints and footprint libraries. The library manager is split into two windows: the library manager window and the footprint editor. The windows open simultaneously, and are tiled vertically.

In the library manager window, you can browse to select the libraries you want to modify during the current session. Once you select a library, you have access to all of the footprints in that library. Using the library manager, you can also create custom libraries, create footprints, and save new or modified footprints to the library of your choice.

The footprint editor is the primary window you use when creating and editing footprints. It provides a graphical display

of the footprint and is specifically tailored for the creation and modification of individual footprints.



To open the library manager

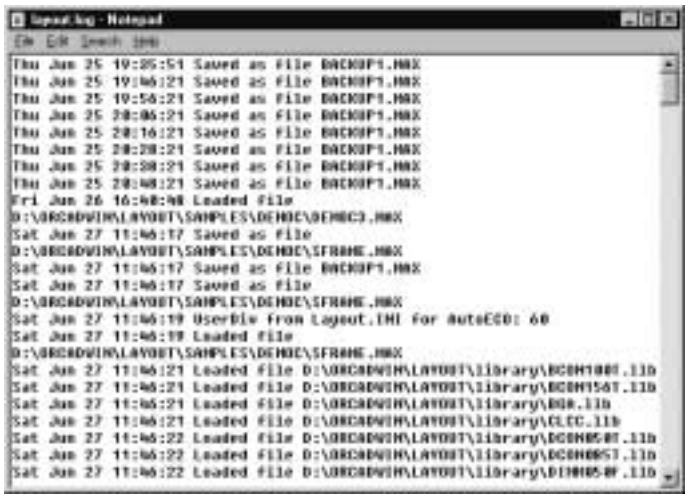
1 Choose the library manager toolbar button or from the File menu, choose Library Manager.

To close the library manager, click on the X in the upper, right-hand corner in either the library manager window or the footprint editor, and choose the OK button when Layout asks if you want to close the library manager.

The session log

The session log lists all the events that have occurred related to the currently open board. If you're experiencing problems with Layout, look in the session log and try to interpret any error messages you see before contacting OrCAD's technical support staff. The information in the session log is useful when

working with the technical support staff to solve technical problems.


layout.log - Notepad
File Edit Search Help
Thu Jun 25 19:05:51 Saved as file BACKUP1.MAX
Thu Jun 25 19:46:21 Saved as file BACKUP1.MAX
Thu Jun 25 19:56:21 Saved as file BACKUP1.MAX
Thu Jun 25 20:06:21 Saved as file BACKUP1.MAX
Thu Jun 25 20:16:21 Saved as file BACKUP1.MAX
Thu Jun 25 20:28:21 Saved as file BACKUP1.MAX
Thu Jun 25 20:38:21 Saved as file BACKUP1.MAX
Thu Jun 25 20:48:21 Saved as file BACKUP1.MAX
Fri Jun 26 16:48:48 Loaded file
D:\ORCADWIN\LAYOUT\SAMPLES\DEMOC\DEMOC3.MAX
Sat Jun 27 11:46:17 Saved as file
D:\ORCADWIN\LAYOUT\SAMPLES\DEMOC\SFRAME.MAX
Sat Jun 27 11:46:17 Saved as file BACKUP1.MAX
Sat Jun 27 11:46:17 Saved as file
D:\ORCADWIN\LAYOUT\SAMPLES\DEMOC\SFRAME.MAX
Sat Jun 27 11:46:19 UserDir from Layout.INI for AutoECO: 60
Sat Jun 27 11:46:19 Loaded file
D:\ORCADWIN\LAYOUT\SAMPLES\DEMOC\SFRAME.MAX
Sat Jun 27 11:46:21 Loaded file D:\ORCADWIN\LAYOUT\library\BGDM100T.lib
Sat Jun 27 11:46:21 Loaded file D:\ORCADWIN\LAYOUT\library\BGDM150T.lib
Sat Jun 27 11:46:21 Loaded file D:\ORCADWIN\LAYOUT\library\BGA.lib
Sat Jun 27 11:46:21 Loaded file D:\ORCADWIN\LAYOUT\library\CLCC.lib
Sat Jun 27 11:46:22 Loaded file D:\ORCADWIN\LAYOUT\library\DGDN05BT.lib
Sat Jun 27 11:46:22 Loaded file D:\ORCADWIN\LAYOUT\library\DGDN05ST.lib
Sat Jun 27 11:46:22 Loaded file D:\ORCADWIN\LAYOUT\library\DIDN05BF.lib

To open the session log

1. From the File menu, choose Text Editor. A text editor (such as Notepad) appears.
2. From the text editor's File menu, choose Open. The Open dialog box appears.
3. Change the Files of type to All Files, locate and select LAYOUT.LOG, then choose the Open button. The session log opens in the text editor window.

The toolbar

Note: The same toolbar appears when you're using the library manager, although some buttons are unavailable (and appear dimmed) because they do not apply to the current activity.

Note: To prevent tooltips from displaying, deselect the Show Tooltips option in the User Preferences dialog box (from the Options menu, choose User Preferences).

By choosing a tool in the toolbar, you can quickly perform the most frequent Layout tasks. When you move the pointer over a toolbar button, the button's name appears below the button, in what is referred to as a *tooltip*.

The table below summarizes the functions of the toolbar icons. The functions are described in detail throughout this manual.

Table 3-1 The toolbar

Tool	Name	Description	Equivalent to
	Open	Opens an existing board.	Open command on the File menu
	Save	Saves an existing board.	Save command on the File menu
	Library manager	Opens the library manager.	Library Manager command on the File menu
	Delete	Deletes whatever you have selected.	Delete command on the Edit menu
	Find	Displays the Find Coordinate or Reference Designator dialog box, which you use to search for specific coordinates or reference designators.	Find/Goto command on the Edit menu
	Edit	Displays an appropriate editing dialog box, depending on what you have selected.	Properties command on the Edit menu
	Spreadsheet	Displays a list of the available spreadsheets.	Similar to the Database Spreadsheets command on the View menu
	Zoom in	Magnifies selected areas of the board.	Zoom In command on the View menu

Table 3-1 The toolbar, *continued*

	Zoom out	De-magnifies selected areas of the board.	Zoom Out command on the View menu
	Zoom all	Zooms so that you can see the entire board.	Zoom All command on the View menu
	Query	Displays the query window, which lists an object's properties.	Query Window command on the View menu
	Component	Enables you to select, add, move, edit, or delete components.	Equivalent to choosing Component, then Select Tool from the Tool menu
	Pin	Enables you to select, add, move, edit, or delete pins.	Equivalent to choosing Pin, then Select Tool from the Tool menu
	Obstacle	Enables you to select, add, move, edit, or delete obstacles.	Equivalent to choosing Obstacle, then Select Tool from the Tool menu
	Text	Enables you to select, add, move, edit, or delete text.	Equivalent to choosing Text, then Select Tool from the Tool menu
	Connection	Enables you to select, add, combine, or delete net connections.	Equivalent to choosing Connection, then Select Tool from the Tool menu
	Error	Enables you to select error markers related to spacing and design rule violations.	Equivalent to choosing Error, then Select Tool from the Tool menu
	Color	Displays the Color spreadsheet, within which you change the color of layers or objects, or their visibility (visible or invisible).	Colors command on the Options menu.

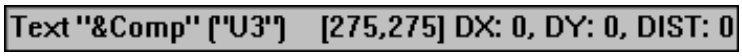
Table 3-1 The toolbar, *continued*

	Online DRC	Enables online design rule checking. The state of online DRC can be viewed in the design window's title bar, which reads either DRC ON or DRC OFF.	Activate Online DRC option in the User Preferences dialog box
	Reconnect	Enables reconnect mode, which you use to show or hide routes and connections. Unlike earlier versions of Layout, reconnect mode should only be used during component placement, before any routing is done.	Instantaneous Reconnection Mode option in the User Preferences dialog box
	Auto path route	Enables auto path route mode (not available in Layout Engineer's Edition), which you use to route and place vias interactively using the shove algorithm.	Auto Path Route Mode option in the Route Settings dialog box
	Shove track	Enables shove track mode, which you use to route manually using the shove algorithm.	Shove Track Mode option in the Route Settings dialog box
	Edit segment	Enables edit segment mode, which you use to select existing tracks and change their positions, while Layout automatically adjusts the angles and sizes of adjacent segments to maintain connectivity.	Edit Segment Mode option in the Route Settings dialog box
	Add/edit route	Enables add/edit route mode, which you use to route manually without using the shove algorithm.	Add/Edit Route Mode option in the Route Settings dialog box
	Refresh all	Minimizes connections, repours copper, and recalculates board statistics.	Equivalent to choosing Refresh, then All from the Auto menu.
	Design rule check	Runs a design rule check using the options selected in the Check Design Rules dialog box (accessed by choosing Design Rule Check from the Auto menu).	OK button in the Check Design Rules dialog box

Table 3-1 The toolbar, *continued*

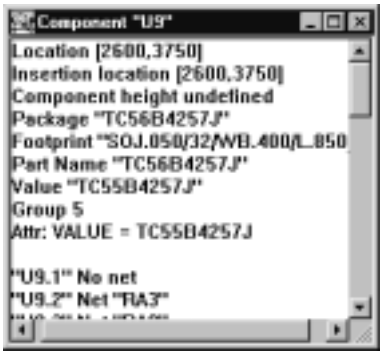
	Selection Filter	Enables selection filter mode, which lets you to select objects on a very dense layout board using keyboard	Activate Selection Filter option in the User Preferences dialog box

The status bar



The status bar is located at the bottom of the design window. It displays the cursor coordinates and system memory. When you select a component, obstacle, pin, text, or track, the status bar displays its name and type. As you move the selected object, the status bar updates coordinates, its distance from its original location, and other relevant information, such as its angle.

Query window



The query window provides detailed data for an object selected in either the design window or in a spreadsheet. When you click on a keyword (marked with quotation marks) in the query window, information about that item appears in the query window, and the item is highlighted on the board.

If you click on a location (the X and Y coordinates given in brackets) in the query window, the location is highlighted on the board and marked with an “X.”

By placing the query cursor (shaped like a Q) in the query window and pressing the ENTER key, an appropriate edit dialog box appears, so that you can edit data. By placing the query cursor in the query window and pressing the TAB key, an appropriate search dialog box (the Find and Select Item dialog box or the Find Coordinate or Reference Designator dialog box) appears.

If you enter the name of an object and choose the OK button in the search dialog box, the information about the object appears in the query window and the object is highlighted on the board.

To open the query window

1 Choose the query toolbar button or from the View menu, choose Query Window.

Using Refresh Hot Link to query spreadsheets

If you open a spreadsheet and choose Refresh Hot Link from its Query window, any objects in the spreadsheet that are related to the object visible in the query window are highlighted on the board and in related spreadsheets. For instance, if you open both the Nets and Components spreadsheets and highlight GND in the Nets spreadsheet, its information appears in the query window, and the components attached to GND are highlighted in the Components spreadsheet.

Pop-up menu

You can display pop-up menus in the design window, library manager, and spreadsheets by pressing the right mouse button. Their commands are specific to the tool you are using. For instance, if you display a pop-up menu in the Design Window with a component selected, the commands will be different than if you had no component selected.

To display Layout's context-sensitive, pop-up menus, you can:

- Click the right mouse button

  or

- Press the plus (+) key on the numeric keypad.

Selecting and deselecting objects

This section describes the different ways to select individual objects and groups of objects. These selection methods work both in the design window and library manager.

There are three selection modes available in Layout:

- autotool
- tool-specific
- block tool

Besides these, there is another selection mode, selection filter mode, which is available only for the design window. This is an alternate selection mechanism provided by Layout and is useful for selecting objects on a very dense Layout board. In this mode the object selection depends on the distance between the object and the cursor position. An object of the enabled mode and nearest to the cursor can be selected using shortcut keys.

Note: You can either use the selection filter or the autotool select mode. If you select the Activate AutoTool Select Mode, the Activate Selection Filter check box inside User Preferences dialog box the will be disabled and vice versa.

When you select the Activate AutoTool Select Mode option in the User Preferences dialog box, Layout selects objects without regard to the *active* tool. The active tool is the tool that you last selected for use. For example, if you last chose the component tool, it is the active tool.

If you have trouble selecting an object using autotool select, it may be too close to surrounding objects. Choose the

appropriate tool before selecting the object. After selecting the object, Layout automatically returns to autotool select mode.

Note: If you pick up the correct object, but on the wrong layer, you can type the layer number for the appropriate layer.

If you don't select the Activate AutoTool Select Mode option, Layout uses the tool-specific method. This method of selection is useful if the board is dense and you have trouble isolating an object using the autotool select mode.

Note: For information on setting user preferences, see *Setting environment preferences*.

To select multiple types of objects (for example, components, text and pins) use the Block tool. This tool allows you to select a rectangular or polygonal area of the board. This tool is very helpful when you want to rotate the entire board file.

To select an object in autotool select mode

1 Click an object with the left mouse button.

To select multiple objects in autotool select mode

1 Press the CTRL key and select each object.

Note: In the design window, pins and error markers cannot be selected using autotool select mode. Hence, you must choose the pin tool or error tool first. However, in the library manager, pins can be selected using autotool select mode, but components cannot.

The reasoning behind this is that, in general, you select a pin in the footprint library, not an entire footprint. If you need to select an entire footprint, choose the component tool first.

To select an object using tools

1 Choose the appropriate tool for the object you want to select.

2 Press the CTRL key and click the left mouse button with the pointer over the object. The selected object appears in the highlight color specified in the Color spreadsheet.

To select multiple objects using tools

1 Press the CTRL key and select each object
or
hold the left mouse button while dragging the mouse, drawing a rectangle around the object or objects to select. Release the left mouse button.

Note: If you want to select an object without moving it, press the CTRL key and click the object with the left mouse button.

2 The selected objects appear in the highlight color specified in the Color spreadsheet.

To deselect objects

1 Press the ESC key or click on an area where there are no objects. If you made the selection with the Block tool, you can deselect individual objects by holding down the CTRL key and clicking each object.

To select a design window object to be moved

1 In the design window, select the appropriate tool.

2 Position the pointer over the object.

3 Click the left mouse button.

To select a design window object without moving it

1 In the design window, select the appropriate tool.

2 Position the pointer over the object.

3 Press the SHIFT key while you click the left mouse button.

To select a design window object with Activate AutoTool Select Mode active

1. From the Options menu, choose the User Preferences command.
2. Select the Activate AutoTool Select Mode option.
3. Choose OK.
4. Position the pointer over the object and click the left mouse button. Layout selects the object and activates the appropriate tool.

   - ❑ If you're selecting a component, click on a pin of the component.
   - ❑ If you're selecting a net or a connection, click on the track.
   - ❑ If you're selecting an obstacle, click on the line that defines the obstacle, rather than within the obstacle area.
   - ❑ If you're selecting text, click on the text.

To select a pin in the footprint editor window

1. In the User Preferences dialog box, select the Activate AutoTool Select Mode option and choose the OK button.
2. Position the pointer over the pin and click the left mouse button.

   To select a group of pins in the Footprint Editor, do one of the following:

   - ❑ Press the left mouse button and drag it over the pins you want to include in the selection
   - ❑ Press the CTRL key and click on the pins you want to include in the selection.

   To deselect a pin in the selection, press the CTRL key and click on the pin.

To select a component in the footprint editor window

1 Activate the Component tool.

2 Position the pointer over the component and click the left mouse button.

To select multiple objects

1 In the design window, select the appropriate tool.

2 Position the pointer over the first object and click the left mouse button.

3 In succession, position the pointer over each additional object and press the CTRL key while you click the left mouse button.

Note: To select and manipulate an area of the board, use the Block tool, as described later in this section.

To select multiple objects for editing with Auto Select active

1 From the Options menu, choose the User Preferences command.

2 Select the Activate AutoTool Select Mode option, then choose the OK button.

3 Press the CTRL key, position the pointer over the first object, and click the left mouse button.

4 In succession, press the CTRL key, position the pointer each additional object, and click the left mouse button.

To deselect objects

1 Position the pointer in an area where there are no objects and click the left mouse button.

To start the Selection Filter mode

1. From the Options drop-down menu choose User Preferences.
2. In the User Preference dialog box, select the Activate Selection Filter check box.

   Note: As user selects the Activate Selection Filter check box inside User Preferences dialog box the Activate AutoTool Select Mode will be disabled and vice versa.
3. If you want the save your modifications in the `layout.ini` file, click the Save User Preferences button.

   The current status for Active Selection Filter check box will be reflected in the LAYOUT.INI file.
4. Click OK.

To select a design window object using Selection Filter

Using Selection Filter, you select objects that are closest to the current cursor position on the layout board. Objects that are close enough to the point of mouse click get added to selection list.

In the Selection Filter mode, two shortcut keys become active. You use these shortcut keys to select the objects and also to change the object selection mode. The shortcut keys available are:

- Press CTRL + Y keys to select the next object from the list in the same mode.
- Press CTRL + H keys to change the current mode of selecting objects.

  The order of change is similar to the appearance of the mode icons in the tool bar, in a cyclic order. This implies that from the Component selection mode, you can switch to Obstacle selection mode, followed by Text selection mode, Error selection mode, Edit Segment mode, and finally, the Add\Edit Segment mode.

If none of the objects of the current mode are close enough to the current cursor position, then no object will be selected. But object of other modes that are near to the cursor will be added to the list. So user can change modes and select objects of other modes by pressing the hot key.

The lists will hold their data until the next time user clicks for selection.

Important

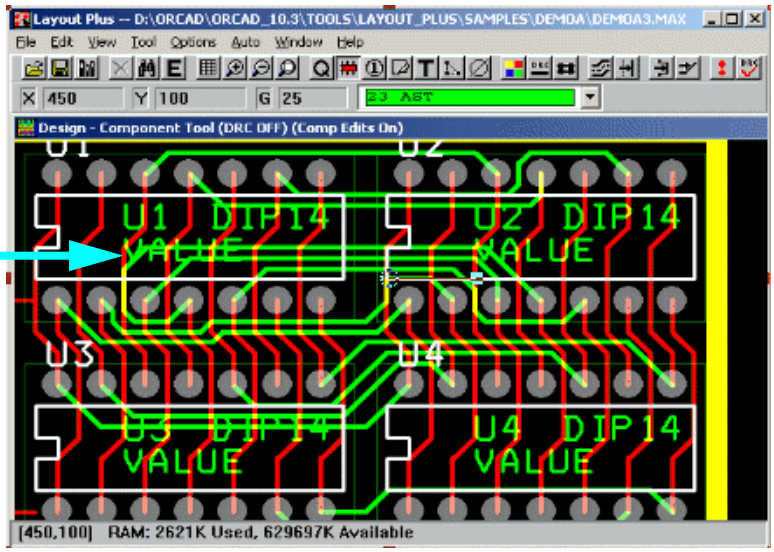
After enabling the selecting filter, if you add or delete components, text, obstacles or errors, or add or unroute segments, connections or nets, you must click on the Layout board again to update the list of objects in the selection filter list.

Note: Selection Filter cannot be used to select free vias and clusters.

Example of using Selection Filter for selecting objects

In this section we will use the Selection Filter to select a text on the Layout board. In a layout board shown below, select the text VALUE.

Text to be selected


Layout Plus -- D:\ORCAD\ORCAD_10.3\TOOLS\LAYOUT_PLUS\SAMPLES\DEMOA\DEMOA3.MAX
File Edit View Tool Options Auto Window Help
X 450 Y 100 G 25
Design - Component Tool (DRC OFF) (Comp Edits On)
U1 DIP14 VALUE
U2 DIP14 VALUE
U3
U4
U3 DIP14 VALUE
U4 DIP14 VALUE
[450,100] RAM: 2621K Used, 629697K Available

1 From the Options drop-down menu select User Preferences.

2 In the User Preferences dialog box, select the Activate Selection Filter check box.

3 Click the Save User Preferences button.

4 Click OK.

Note that the cursor would change to cross-hair and the Component Tool button is selected in the toolbar, indicating that you are in the Selection Filter mode.

5 Click on the component U1.

U1 will get attached to the cursor.

6 To move to the text selection mode, press CTRL + H keys.

Obstacle Tool gets selected.

7 To select the Text Tool, again press CTRL + H keys.

The Text Tool gets selected in the toolbar.

8 Press CTRL + Y keys.

The reference designator U1 gets attached to the cursor.

9 Again press CTRL + Y keys.

The text VALUE is now attached to the cursor. You can now move or edit the text value.

10 To move out of the selection filter mode, right-click and from the pop-up menu choose End Command.

To change the highlight color

1 In the design window, choose the Color toolbar button.

2 In the Color spreadsheet, double-click in the Highlight (Any layer) row.

3 In the Edit Color dialog box, select a color, then choose the OK button.

To select data in a spreadsheet

- To select a cell, position the pointer in the cell and simply click the left mouse button in that cell.
- To select a row, click the left mouse button in the leftmost cell of that row.
- To select a column, click the left mouse button in the topmost cell of that column.

To select and move a rectangular block of the design with the Block tool

1. From the Tool menu, point to Block and choose Select Tool.
2. Click and drag to create a rectangular selection. The selected components, tracks and vias highlight.
3. Right-click on the selection and choose Move On/Off from the Query window. The selection is attached to the pointer.

   Note: You can temporarily drop a selection by choosing Move On/Off from the pop-up menu.
4. Move the selection to the desired location and click to place.

To select and move polygonal block of the design with the Block tool

1. From the Tool menu, point to Block and choose Select Tool.
2. Click each vertex of the polygonal selection.
3. When finished defining the polygon, right-click and choose Finish from the pop-up menu. The selected components, tracks and vias highlight.

   Note: You can also finish a selection by double-clicking. This places the final vertex and finishes the selection.

4 Right-click on the selection and choose Move On/Off from the pop-up menu. The selection is attached to the pointer.

Note: You can temporarily drop a selection by choosing Move On/Off from the pop-up menu.

5 Move the selection to the desired location and click to place.

To rotate the entire board

1 From the Tool menu point to Block and choose Select All. All items on the board are selected except for datum, error markers and the drill chart.

Note: You can only use the Select All command if you do not have any other items selected on the board.

2 From the Tool menu point to Block and choose Rotate.

3 Right-click in the design window and choose End Command from the pop-up menu to place the board.

Related topics

User Preferences command

The mouse, pointer and cursor

The mouse, pointer and cursor

The Mouse

You can click the left mouse button or press the SPACEBAR to select objects for moving, and click the left mouse button to place moved objects.

In the design window, you can move an object or text by clicking the left mouse button on the item. The object moves with your pointer until you click the left mouse button to place the item.

In the design window, you can select an object for stationery editing by pressing SHIFT and clicking the left mouse button over the object or positioning the pointer over the object then pressing the SPACEBAR.

To select an area, press and hold the left mouse button while you drag a selection rectangle around the item or items you wish to select.

To select noncontiguous graphical objects or spreadsheet cells, hold down SHIFT and click the left mouse button on each object.

In a spreadsheet editor, click the left mouse button in a cell to select a single cell, or click in the leftmost column to select the row, or click the column head to select the column, then click the appropriate mouse button to display a context-sensitive, Query window for editing the cell or cells.

If you do not have a mouse, or prefer to use the keyboard, you can open any menu by typing ALT+ the key that is underlined on the menu bar, then typing the letter of the option that is underlined on the pull-down menu.

Pressing the plus (+) key on the numeric keypad or pressing the right mouse button displays a context-sensitive pop-up menu.

For further information on using a mouse in a Windows environment, refer to your Windows documentation.

The pointer and cursor

In Layout, there are several types of pointer and cursor indicators.

- An arrow indicates that menu commands such as File, Save, or Zoom are accessible.
- An hourglass indicates that the computer is busy and no functions are accessible.

- A text cursor (an elongated "I") indicates that the computer is waiting for you to make an entry in the text entry box in a dialog box.
- A large cross (about 3/8" in diameter) indicates that the computer is waiting for you to select an item or issue a command. It is seen inside an active routing window.
- A small cross (about 1/8" in diameter) indicates that you are actively dragging a route inside the routing window, or that the obstacle tool is waiting for you to begin drawing a new obstacle.
- A zoom pointer (a "Z") indicates that one of the Zoom commands is active.

Density Graph window

The Density Graph shows the relative connection density of the board, so that you can examine the routing difficulty associated with various placement options. Layout considers a number of factors (number of routing layers and surface mount parts, track widths, and DRC settings) to determine the connection density of the various areas of your design. The color that appears in a particular area of the window indicates its connection density:

- **Black**: no appreciable connections
- **Blue**: Less than 25% connection density
- **Green**: Less than 50% connection density
- **Yellow**: Less than 75% connection density
- **Pink**: Greater than 75% connection density
- **Red**: Greater than 100% connection density

The bar graphs at the top and right display the density by direction. That is, the bar graphs at the top of the design reflect the density for vertical routes; the bar graphs to the right of the design reflect the density for horizontal routes.

As you place your components in the design window, use the density graph to determine how your placement affects the

potential success of a routing operation. That is, place your components in the design window, then consult the density graph to determine the connection density for that placement. Return to the design window and adjust your placement accordingly, then check the density graph again. Repeat the process until you are satisfied that connection density is at an acceptable level. In general, it is best to place your components such that there are no areas with more than 75% connection density (there are no pink or red areas) if possible.

There are three levels of detail available in the density graph: Coarse, Medium and Fine. In the initial placement, use the Density graph at the Coarse level and place your components such that the connection density meets your requirements. Then, set the Density graph to Medium, and further identify problem areas in your placement. Once you reduce the connection density at the Medium level, move to Fine. It may not always be possible to eliminate all high-density connection areas. However, if you are able to identify these areas with the Density graph, it will allow you to develop an appropriate routing strategy. (As a general rule, you should route high-density areas first.)

Shortcut

Keyboard: SHIFT+H

Related topics

Database Spreadsheets command (View menu)

Coarse command

Medium command

Fine command

Strategy command (Spreadsheets)

The design window

The library manager

Viewing the current coordinates

The X and Y coordinates corresponding to the location of the cursor appear below the toolbar buttons. The value is measured in the units of measurement you specify in the System Settings dialog box, accessed by choosing System Settings from the Options menu.

Viewing the place grid

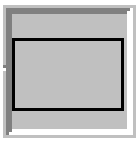
The current place grid setting appears directly below the toolbar buttons. The display reflects the Place grid value you specify in the System Settings dialog box, accessed by choosing System Settings from the Options menu in the design window.

Viewing the current layer

The active board layer and its color appear below the toolbar buttons in the layer drop-down list. You can change layers by choosing one from the list, or by typing the number corresponding to the layer you want. For example, type 1 to change to the top layer.

Note: If you want to view just one layer, press the BACKSPACE key to clear the screen, then type the layer number. Note that actions you perform may affect all layers, even though only one is visible. Pressing the HOME key redraws all layers.

Using the postage stamp view



A miniature outline of the board appears at the far right of the toolbar buttons. You can use this to determine what your current view is in relation to the entire board. You can change the view by moving your cursor into the *postage stamp* view and clicking on a different area. Or, you can draw a window within the postage stamp view to zoom to that window. Double-clicking in the postage stamp view has the same effect as choosing Zoom All from the View menu.

The Help System

The following commands are on the Help menu:

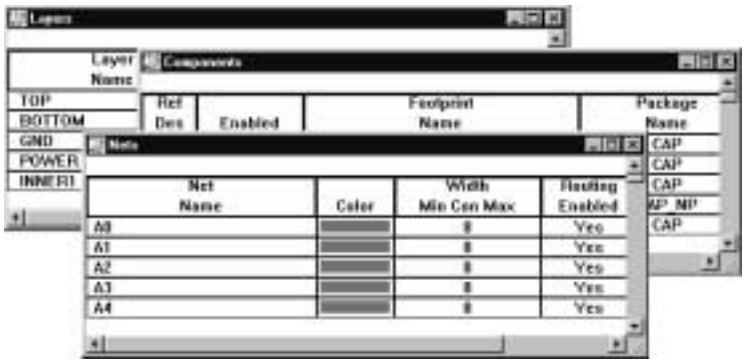
- **Layout Help** - Displays the Help contents page, which lists general categories of information you can browse through, including Layout features, commands, and dialog boxes.
- **What's New** - Displays the What's New document for this release of Layout. What's New documents contain a list of current enhancements and bug fixes for this release of Layout.
- **Known Problems and Solutions** - Opens the Known Problems and Solutions document. Descriptions of identified problems and the associated Cadence Product Change Request (PCR) numbers are listed.
- **Web Resources -** Links to Layout resources on the web
- **Learning Layout** - Displays the online, interactive tutorial.
- **Documentation** - Displays the Cadence Online Documentation system. Through this system you can access the manuals for all installed Cadence products.

- **About Layout** - Displays the Layout version number, licensing information, and your registration number.

Spreadsheets

Layout provides a variety of spreadsheets that you can use to view and edit board information. To display most of the spreadsheets, choose the spreadsheet toolbar button, then choose a spreadsheet. Or, choose Database Spreadsheets from the View menu and choose a spreadsheet.

Note: If you want to select every element in a spreadsheet, click in the left most column's title cell.



Because the routing-related spreadsheets are used in setting routing strategies, you can display them by choosing the spreadsheet toolbar button, choosing Strategy, then choosing a spreadsheet. Alternatively, from the Options menu, choose Route Strategies, then choose a spreadsheet. From the Options menu, choose Global Spacing to display the Route Spacing spreadsheet.

Because the placement-related spreadsheet is used to set autoplacement strategy, you can display it by choosing the spreadsheet toolbar button, choosing Strategy, then choosing Place Pass. Alternatively, from the Options menu, choose Placement Strategy.

Note: The Place Pass spreadsheet is only available in Layout Plus.

From the Options menu, choose Colors to display the Color spreadsheet, or choose Post Process Settings to display the Post Process spreadsheet.

Route Sweep spreadsheet

Use the Route Sweep spreadsheet to view the settings (routing window size, overlap percent, and sweep direction) for the six main routing sweeps Layout uses to try to route a board to 100%.

Route Pass spreadsheet

Use the Route Pass spreadsheet to view the routing strategies (via cost, retry cost, route limit, and attempts) and routing algorithms (heuristics, maze, Auto DFM, fanout, via reduce, and Auto CDE) Layout uses in its routing passes.

Route Layer spreadsheet

Use the Route Layer spreadsheet to view whether a layer is enabled for routing, the primary direction of a layer, its layer cost (a low cost for a layer indicates that the layer is preferred for routing), and its between pins cost (the cost of routing between pins on 0.100 (or less) centers).

Route Spacing spreadsheet

Use the Route Spacing spreadsheet to view the settings for the various spacing criteria (track to track, track to via, track to pad, via to via, via to pad, and pad to pad) Layout uses when routing and when checking for DRC violations.

Editing spreadsheet information

Layout's spreadsheets not only visually and structurally organize the information and elements that comprise your board, they also provide a means for editing board data.

There are two ways to edit board data using the spreadsheets. You can access dialog boxes by double-clicking in a spreadsheet. Or, you can access a Query window by pressing the right mouse button while in a spreadsheet.

Note: If you select multiple rows in a spreadsheet and try to edit them, you may find some of the options in the editing dialog box for that spreadsheet are grayed out. Interpret this to mean that Layout can't tell you the state of the item.

To edit spreadsheet data

1. Choose the spreadsheet toolbar button and choose a spreadsheet.
2. Do one or more of the following:
   - **a.** Double-click in a cell to open a dialog box with that cell's information available and the other cell's information unavailable (dimmed).
   - **b.** Double-click in a column heading to open a dialog box with the column's information available and other information unavailable (dimmed).
   - **c.** Double-click in the first cell of a row to open a dialog box with all of the editable options for that row available.
   - **d.** Double-click in the first column's heading to open a dialog box with all of the editable options for all of the rows of the spreadsheet available.
   - **e.** Press the right mouse button to display a Query window, then choose one of the commands.

Statistics spreadsheet

The Statistics spreadsheet provides basic information about the number of objects in the current design, rate of completion (for placement and routing), and board status. The Enabled column reports the components and nets that are active. The Total column reports the enabled components and nets plus

any disabled components and nets. Statistics can be in inches or centimeters. The default is to display in inches.

Shortcut

Keyboard: SHIFT+I

To view the board statistics in centimeters

1. From the Options menu, choose System Settings. The System Settings dialog box appears.
2. Select one of the metric modes (Centimeters, Millimeters, or Microns).
3. Press OK.
4. From the toolbar, select the Spreadsheets button and choose Statistics. Layout refreshes the screen and displays the design using the new units.

The Statistics spreadsheet contains two columns of information:

- **Enabled -** Includes routing-enabled nets and components only.
- **Total -** Includes information on all nets and components, whether routing-enabled or not.

Related topics

System Settings dialog box

Database Spreadsheets command (View menu)

Strategy command (Spreadsheets)

Layers spreadsheet

Use the Layers spreadsheet to view, add, disable, or modify the board layers. The Layers spreadsheet provides

information on layer names, layer nicknames, layer types, and mirror layers.

Shortcut

Keyboard: SHIFT+Y

You can use the Edit Layer dialog box to assign mirror layers, so that Layout will automatically mirror components, obstacles and text, reversing pad stacks and component outlines accordingly. You can also use the Edit Layer dialog box to designate a Layer Library Name, so that Layout knows the layer of the library from which to take the padstacks, obstacles, and text.

To display the Edit Layer dialog box

1 Double-click on an item in the spreadsheet.

While the spreadsheet is displayed, there are also editing commands available from the Edit menu, as well as from the Query window.

Related topics

Database Spreadsheets command (View menu)

Edit Layer dialog box

Strategy command (Spreadsheets)

Padstacks spreadsheet

Use the Padstacks spreadsheet to view and edit the location, type, and size of pads. Each padstack has a name, slightly offset from the layer definitions, and a size defined for each layer. Plane layer padstack sizes define clearance.The Padstacks spreadsheet provides information on the shape, size, and offset for each of the vias and padstacks defined for your board.

To view the Padstacks spreadsheet, select the spreadsheet tool, then choose Padstacks.

Shortcut

Keyboard: SHIFT+T

You can use the Edit Padstack dialog box to edit a padstack or via. You can use the Edit Padstack Layer dialog box to edit the definition for a single layer.

Each OrCAD-provided technology template defines seven padstacks for use in your board. These padstacks are designed to meet the needs of the specified board type. You can add padstacks by making a copy of one of the seven, then editing the copy to suit your purposes.

To display the Edit Padstack dialog box

1 In the spreadsheet, double-click on the via name or padstack name.

To display the Edit Padstack Layer dialog box

1 In the spreadsheet, double-click on the layer name for the via or padstack that you want to edit.

While the spreadsheet is active, there are also editing commands available from the Edit menu, as well as from the Query window.

Note: Padstack and via names must not contain spaces.

Note: In Layout, you can establish up to forty-six drill sizes on a board.

Related topics

Database Spreadsheets command (View menu)

Strategy command (Spreadsheets)

Footprints spreadsheet

Use the Footprints spreadsheet to view, access, and edit the library of physical parts used in the board. The Footprints spreadsheet provides information on the pad locations and padstacks used for each footprint in the design, routing under pads, and legal routing pad exit directions.f

From the spreadsheet, you can access the Edit Footprint and Edit Pad dialog boxes. Changes you make in the Edit Pad dialog box apply only to the selected pad, while those you make in the Edit Footprint dialog box apply to the footprint and to its pads. In either dialog box, you can change the route entry/exit rules. You use the Edit Pad dialog box to change the pad name. Any available padstack can be assigned to a pin.

To display the Edit Footprint dialog box, double-click on a footprint row of the spreadsheet. To display the Edit Pad dialog box, double-click on the leftmost column of the row for the pad you want to alter: As with any other spreadsheet, you can change the contents of a particular cell by double-clicking in that cell.

While the spreadsheet is displayed, there are also editing commands available from the Edit menu, as well as from the Query window.

You can see the graphic representation of footprints in the Library Editor.

Shortcut

Keyboard: SHIFT+F

Related topics

Database Spreadsheets command (View menu)

Edit Footprint dialog box

Packages spreadsheets

Use the Packages spreadsheet to view and edit the logical gate and pin information for gate and pin swapping. The Packages spreadsheet provides the electronic gate and pin information that is associated with the components in the design.

Shortcut

Keyboard: SHIFT+K

You can use the Package Edit dialog box to control gate swapping (between identical components or within a component), pin swapping, and gate arrangement within a part.

To display the Package Edit dialog box

1 Double-click on an item in the spreadsheet.

While the spreadsheet is displayed, there are also editing commands available from the Edit menu, as well as from the Query window.

Headings

- **Package name**. A text string designating the electrical package.
- **Gate Name**. Usually an alphabetic character that designates the gate to which each pin belongs. Each gate in a package must have a unique gate name, and all pins in a single gate must share the gate name.
- **Pin Name**. Identifies each pin in terms of its electrical characteristics. Each pin with a gate must have a unique identifier.
- **Gate Group**. An integer. All gates assigned to the same Gate Group can be swapped. Gate Group 0 is a special case representing non-swappable gates.

- **Pin Group**. An integer. All pins assigned to the same Pin Group can be swapped. Pin Group 0 is a special case representing non-swappable pins.
- **Pin Type**. This value can be None, Source, Load, or Terminator. Standard TTL -type pins are usually set to None to indicate that the pin is not a source, load, or terminator.

Related topics

Database Spreadsheets command (View menu)

Components spreadsheet

Use the Components spreadsheet to view and edit the component footprint, package name, location, rotation, routing status, and group. The Components spreadsheet provides information on the routing status, associated footprint shape, and location of each component on a board.

Shortcut

Keyboard: SHIFT+C

You can use the Edit Component dialog box to select or deselect specific components for routing purposes.

To display the Edit Component dialog box

1 Double-click on an item in the spreadsheet.

While the spreadsheet is displayed, there are also editing commands available from the Edit menu, as well as from the Query window.

Related topics

Database Spreadsheets command (View menu)

Strategy command (Spreadsheets)

Query Window command (View menu)

Nets spreadsheet

Use the Nets spreadsheet to set net properties such as width, route enabling, plane layer enabling, and shove. These properties affect both manual and automatic routing. The Nets spreadsheet provides information on signal names, track widths, and routing attributes that have been assigned to the nets in the design. A net is defined as a set of point-to-point electrical connections that have been assigned a net name. There may be multiple connections per net.

Shortcut

Keyboard: SHIFT+N

You can use the Edit Net dialog box to change track widths for nets, enable or disable the routing of specific nets, set the priority (weight) for the routing of each net, set the reconnect rules per net, and set other routing flags, such as Shove Enable, Retry Enable, Copper Share Enable, or Reconnect Enable.

To display the Edit Net dialog box

1 Double-click on an item in the spreadsheet.

While the spreadsheet is displayed, there are also editing commands available from the Edit menu, as well as from the Query window.

Related topics

Database Spreadsheets command (View menu)

Edit Net dialog box

Obstacles spreadsheet

Use the Obstacles spreadsheet to view and edit the obstacles you create, including assembly drawings, silkscreens, copper pour zones, and board outlines. The Obstacles spreadsheet provides information on electrical and non-electrical items such as place outlines, copper spots, via keepouts, and other routing obstacles in the design. It also shows the board outline, which is on layer 0. (There is only one board outline in each design.)

Shortcut

Keyboard: SHIFT+O

You can use the Edit Obstacle dialog box to change obstacle width or height, obstacle type, and layer.

To display the Edit Obstacle dialog box

1 Double-click on an item in the spreadsheet.

While the spreadsheet is displayed, there are also editing commands available from the Edit menu, as well as from the Query window.

Related topics

Database Spreadsheets command (View menu)

Edit Obstacle dialog box

Text spreadsheet

Use the Text spreadsheet to view and edit board text. The Text spreadsheet provides information (such as text name, text string, and text type) for all text in the design.

Shortcut

Keyboard: SHIFT+X

You can edit any text item in this spreadsheet using the Text Edit dialog box.

To display the Text Edit dialog box

- Double-click on an item in the spreadsheet.

While the spreadsheet is displayed, there are also editing commands available from the Edit menu, as well as from the Query window.

Related topics

Text Edit dialog box

Database Spreadsheets command (View menu)

Strategy command (Spreadsheets)

Error Markers spreadsheet

Use the Error Markers spreadsheet to view error types and error marker locations. You can delete error markers from the board by deleting them in the spreadsheet. The Error Markers spreadsheet lists all errors in the design file that are found by the error-checking routines in Layout.

Shortcut

Keyboard: SHIFT+M

While the spreadsheet is displayed, there are also editing commands available from the Edit menu, as well as from the Query window.

Related topics

Database Spreadsheets command (View menu)

Design Rule Check command

DRC/Route Box command (Delete Violating Tracks)

Drills spreadsheet

Use the Drills spreadsheet to view and edit drill sizes, symbols, and tolerance. The Drills spreadsheet provides information on the parameters pertaining to the automatic generation of a drill chart. You can select the symbol used for a particular drill size and can input tolerance and comment text. Each drill entry must have a unique symbol assigned to it.

Shortcut

Keyboard: SHIFT+R

You can use the Drill Properties dialog box to change a symbol number or letter, to enter or change the drill tolerance, and to enter or change drill note text.

To display the Drill Properties dialog box

1 Double-click on an item in the spreadsheet.

While the spreadsheet is displayed, there are also editing commands available from the Edit menu, as well as from the Query window.

Related topics

Drill Properties dialog box

Drill Chart Properties dialog box

Apertures spreadsheet

Use the Apertures spreadsheet to view and edit D-codes and their widths, heights, and shapes. The Apertures spreadsheet provides the width (or outer dimension for thermal relief and annular shapes), height (or inner dimension for thermal relief shapes), and shape of each D-code associated with the apertures in the design.

Shortcut

Keyboard: SHIFT+P

You can use the Edit Apertures dialog box to change the D-code, width, outer dimension, height, or inner dimension of apertures.

To display the Edit Apertures dialog box

- Double-click on an item in the spreadsheet.

Pop-up menu

The Query window for the Apertures spreadsheet has the following options:

- **End Command** - Clears any selections you made in the spreadsheet.
- **New** - Displays the Edit Apertures dialog box so you can create new aperture definitions.
- **Read GerbTool apertures** - Displays the Input Aperture List dialog box, in which you specify a filename with an .APP extension. The .APP file's information is added to the Apertures spreadsheet.
- **Write GerbTool apertures** - Displays the Output Aperture List dialog box, in which you specify a filename with an .APP or a .GTD extension. The information in the spreadsheet is saved to the .APP or .GTD file you specify.

- **Write FMT apertures** - Displays the Output Aperture List dialog box, in which you specify a filename with an .FMT extension. The information in the spreadsheet is saved to the .FMT file you specify.

Related topics

Database Spreadsheets command (View menu)

Strategy command (Spreadsheets)

Query Window command (View menu)

Color spreadsheet

The Colors command, or the Color toolbar button displays the Color spreadsheet. Changes you make to the Color spreadsheet can be saved as a strategy file, if you want.

You can also use the Color spreadsheet to set color and visibility when post processing.

Shortcut

Keyboard: SHIFT+L

Related topics

Color dialog box

Add Color Rule dialog box

Assigning a color to a board layer

Database Spreadsheets command (View menu)

Strategy command (Spreadsheets)

Post Process spreadsheet

Use the Post Process spreadsheet to view and edit the post processing settings for creating Gerber files and for printing or plotting output. Display the Post Process spreadsheet by choosing Post Process Settings from the Options menu, or choose Database Spreadsheets from the View menu, then choose Post Process. To post process a board using Layout, make a list of each output file that you want from the system using the Post Process spreadsheet.

Shortcut

Keyboard: SHIFT+S

Headings

The Post Process spreadsheet is divided into 5 headings:

- **Plot Output File Name** - The name of the output file that is written to disk (in your current directory). This can be a Gerber file or one of many report files, such as a drill list.
- **Enabled (Post Process)** - Lets you enable or disable a particular plot for processing.
- **Device** - Lets you choose between various Layout supported output devices such as Gerber, DXF, HPGL, Laser and Dot Matrix printers.
- **Shift** - Indicates whether the plot is scaled, rotated, or mirrored, and indicates whether the output is shifted or is centered. If it centered, the drill tape coordinates match the coordinates that you see in the design window
- **Plot Title** - Provides a place for you to label each plot. To see what each artwork layer looks like after post processing, select a layer, then select Preview from the Pop-up menu.

Note: You do not need to specify a plotter driver in Layout when the print manager is the selected output device, because Layout uses the plotter driver that you have

selected as your Windows plotter. For additional information on plotters and drivers, see your Windows documentation.

If the print manager is not the selected output device, then you need to select and output a device supplied by Layout (such as Gerber or DXF).

Note: Customers have reported that according to the Product Comparison Guide for HP-GL/2 and HP RTL Peripherals published by Hewlett-Packard, the following devices are obsolete and no longer supported. Therefore, you may be unable to acquire a current, working driver for the following output devices.

- HP DesignJet plotters (except 200, 600, and 650C)
- HP PaintWriter XL printer
- HP DraftMaster Series plotters - SX, RX, and MX (except the SXplus, RXplus, and Mxplus)
- HP 7600 Series plotters - Models 240D/E, 250, 255, and 355

Certain companies (for example, WinLINE) produce drivers. You can contact WinLINE on the internet at http://www.winline.com.

Pop-up menu

The pop-up menu for the Post Process spreadsheet includes the following options:

- **Run Batch** - Generates the various output files for all board layers that are currently enabled.
- **Preview** - Hides the board display and displays a preview of the output file (for example, a Gerber file) for visual inspection.
- **Save Colors** - Retains any color edits you make to the output file. For example, if you change the color of the board outline such that it displays in a Gerber file, you can run the Update Colors command to tell Layout to save this

change of color so that it displays the next time you view the Gerber file.

- **Restore** - Closes the output file preview and restores the board display.
- **Gerber Preferences** - Displays the Gerber Preferences dialog box.

Related topics

Gerber Preferences dialog box

Generate Reports dialog box

Database Spreadsheets command (View menu)

Place Pass spreadsheet

Use the Place Pass spreadsheet to view and edit the settings (iterations, attempts, and maximum clusters) for the six placement operations (assign clusters, proximity place, adjust components, place clusters, swap components, and swap pins) Layout Plus uses during autoplacement.

Note: The Place Pass spreadsheet is only available in Layout Plus.

Editing object properties

Each object has a set of property values that you can edit. Editing properties usually affects the appearance and function of an object.

To edit object properties

1. Select the object.
2. Choose the Properties command from the pop-up menu. An appropriate editing dialog box appears.

3 Change the values as necessary, then choose the OK button.

Setting environment preferences

In Layout, you can edit the default settings that affect your design environment.

To set user preferences

1 From the Options menu, choose User Preferences. The User Preferences dialog box appears.

2 Edit the options to reflect your preferences and requirements, then choose the OK button.

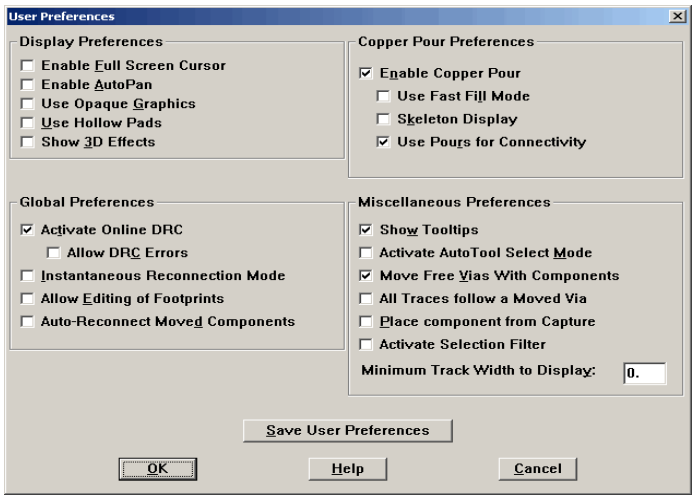


Figure 3-1 User Preferences dialog box

Display Preferences

Enable Full Screen Cursor Changes the cursor to a full screen cursor, with X and Y axes that extend the width and height of the design window.

Enable Auto Pan	With an object selected, placing the pointer at the horizontal edge of the design window causes Layout to pan in the direction of the pointer. The pointer and selected object move to the middle of the screen after panning.
Use Opaque Graphics	When selected, tracks and other objects are solid. You cannot see what, if anything, is under them. When not selected, tracks and other objects are translucent and you can see the tracks and objects beneath them.
Use Hollow Pads	Displays solid pads as hollow squares or circles to reduce redrawing time. They do not print or plot as hollow.
Show 3D Effects	Displays three-dimensional images representing component heights on the screen, and indicates the height on the image. Also displays the identifying text associated with height restrictions or group restrictions for height or group keepins and keepouts.

Global Preferences

Activate Online DRC	Enables online design rule checking. Equivalent to choosing the online DRC toolbar button. With this option selected, you can only draw routes that conform to your space settings.
Instantaneous Reconnection Mode	Enables reconnect mode, which you can use to show or hide nets. Equivalent to choosing the reconnect toolbar button.
Allow Editing of Footprints	Enables you to edit component footprints on the board without opening the library manager. You can edit obstacles, text, and pins attached to separate components.

Copper Pour Preferences

Enable Copper Pour

Enables copper pour drawing and refreshing. You must select this option in order to select the Use Fast Fill Mode option or the Use Pours for Connectivity option.

Note: In the User Preferences dialog box, ensure that the Enable Copper Pour option is selected before you create a Gerber plot. Otherwise, your Gerber plots will have no copper pour in them.

Use Fast Fill Mode

Reduces the drawing time for copper pour by using a simple pattern to represent copper pour on your screen. This option only affects the display of the copper pour on the screen. It does not accelerate the actual pour process.

Use Pours for Connectivity

Layout considers connections to be routed when they exist in a copper pour, provided that the copper pour is common to the same net as the pins.

Miscellaneous Preferences

Show Tooltips

Displays tool descriptions as you pass your cursor over the toolbar buttons. It also enables the use of pop-up dialog boxes as error indicators. If you do not select this option, Layout uses beeps to indicate errors and displays the errors in the status bar.

Activate AutoTool Select Mode

Enables you to select an object without having to choose the appropriate tool first.

Move Free Vias With Components	Enables you to move free via fanouts with components.
All Traces follow a Moved Via	Ensures that the traces attached to a via remain attached to the via when you move the via.
Place component from Capture	Enables you to cross-place components from OrCAD Capture to Layout. When you select a component in Capture and move the cursor over the Layout window, the component is attached to the cursor in a place (or move) mode. For more information on cross-placing components from Capture to Layout, see Cross-placing components from Capture to Layout on page 333.
Activate Selection Filter	Enables you to select objects that are closest to the current cursor position on a densely populated layout board, by using shortcut keys.
Minimum Track Width to Display	Reduces the redraw time for wide tracks by using a minimum width to represent the tracks. Layout draws tracks wider than this setting as actual size, and draws all other tracks as a single pixel line.

Save User Preferences

Save User Preferences Button	Saves the user preferences settings in your local directory. Future Layout sessions use the saved settings.

Using and assigning colors

Layout assigns a default color for each board layer. You can use the Color spreadsheet to edit the colors used in the graphical display and to make layers visible or invisible.

Note: You can save a color scheme as a strategy file for use with future boards. To do so, define the colors using the instructions in this section, then use the Save As command (from the File menu) to save the file with an .SF extension.

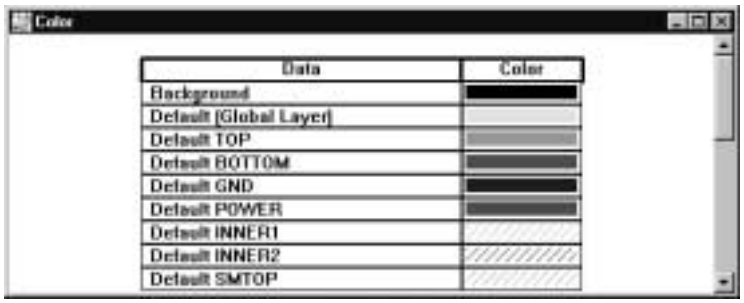
Note: Layout uses a different process for specifying the colors you want to use for preview and output. For information on using color during post processing, see Previewing a layer on page 511.

To open the Color spreadsheet

1 Choose the color toolbar button
or
from the Options menu, choose Colors.

Layout displays the Color spreadsheet.

Note: Diagonal lines within a color box indicate that a layer and the objects on that layer are set to invisible.



To change the color of an object or layer

1 Select an item in the Color spreadsheet.

Note: The ratsnest color is set in the Nets spreadsheet, not in the Color spreadsheet. Routed track color, on the other hand, is set in the Color spreadsheet, and is usually left as the default color for the layer.

2 From the Query window, select a new color for the item or click the Define Custom Colors button to create a custom color.

3 Choose the OK button.

4 Close the Color spreadsheet. The item appears in the new color.

Assigning a color to an object

Objects take on the color of the layer on which they reside unless they are specifically assigned a color in the Color spreadsheet. You can assign a color to all objects of a specific type or you can assign a color to objects of a specific type and on a specific layer.

Objects must be listed in the Color spreadsheet and visible in order to be edited. For instructions on this, see Changing the visibility of a layer or an object.

Note that the ratsnest color is set in the Nets spreadsheet, not in the Color spreadsheet. Routed track color, on the other hand, is set in the Color spreadsheet, and is usually left at the default color for the layer.

The color assignment feature is very powerful, and allows you to assign multiple colors to a single object. If that occurs, the conflict is resolved so that a color assigned to an object on a particular layer has first priority, a color assigned to an object on any layer (the dash) has second priority, and the default color for the layer has the lowest priority.

To assign a color to an object

1 In the design window, choose the Color toolbar button.

2 Verify that the object you want to edit is in the spreadsheet and that it is visible. For instructions, see Changing the visibility of a layer or an object.

3 In the Color spreadsheet, select the object, then from the Query window choose the color.

4 Close the Color spreadsheet to display the design window with the object displayed in the new color.

Note: If you want to change the visibility or the color of a particular object on a particular layer, in the Add Color Rule dialog box you must select the appropriate option button and enter the layer in the edit box.

Related topics

Changing the visibility of a layer or an object

Assigning a color to a board layer

Colors command

Color dialog box

Gerber plot preview

Assigning a color to a board layer

The default color for a layer is the color for all objects on that layer, except where specific objects have been assigned another color. Typically, a default color is assigned for each routable layer of a design.

Note that the Placement strategy files and Routing strategy files already have layer colors assigned, so there is usually little need to customize them to create a color setup for each task in the process of finishing a PCB.

Note also that the ratsnest color is set in the Nets spreadsheet, not in the Color spreadsheet. Routed track color, on the other hand, is set in the Color spreadsheet, and is usually left as the default color for the layer.

To assign a default color to a layer

1 In the design window, choose the Color icon.

2 Verify that the layer you want to edit is in the Color spreadsheet and that it is visible. For instructions, see Changing the visibility of a layer or an object.

3 In the Color spreadsheet, select the layer, then from the Query window choose the color.

4 Close the Color spreadsheet to display the design window. All visible objects on the layer show up in the selected color.

To make a layer visible or invisible

1 Select a layer in the Color spreadsheet.

2 From the pop-up menu, choose Visible<>Invisible. The color appears as a solid color if you made the layer visible, or as a diagonal pattern if you made the layer invisible.

Note: You can also toggle current layer visibility off and on by choosing Visible<>Invisible from the View menu, or by typing a dash (-).

3 Close the Color spreadsheet.

Related topics

Color dialog box

Add Color Rule dialog box

Changing the visibility of a layer or an object

Assigning a color to an object

Gerber plot preview

Changing the visibility of a layer or an object

In Layout, each layer and each category of objects can be set to visible or invisible. You may want to use visibility in order to

control visual clutter in a window or to control what is present in your printed or plotted output.

A layer must be visible for its components to be selectable, regardless of whether you are using single select, area select, or SHIFT + CLICK mode. If the top layer is invisible, top layer components are not selectable. The same is true for bottom layer components if the bottom layer is invisible.You set the visibility from the Color spreadsheet. Objects not listed in the Color spreadsheet cannot be edited, but you can easily add them to the spreadsheet.

In the Color spreadsheet, a solid color block indicates visibility, while a cross hatch color block indicates the object or layer is invisible.

To add a layer or object to the Color spreadsheet.

1. In the design window, choose the Color toolbar button to display the Color spreadsheet.
2. From the Query window, choose the New command.
3. In the Add Color Rule dialog box, select the option button that matches the object. If you are adding a layer, enter the layer nickname, or the layer number as shown in the layer drop-down list of the design window, or the library layer name if you are working in the library manager. You may also use a dash "-" to specify any layer; this signifies any occurrence of the selected object.
4. Choose the OK button. The layer or object is now listed in the Color spreadsheet and available for editing.

Note: If you want to change the visibility or the color of a particular object on a particular layer, in the Add Color Rule dialog box you must select the appropriate option button and enter the layer in the edit box.

To change the visibility of a layer or an object.

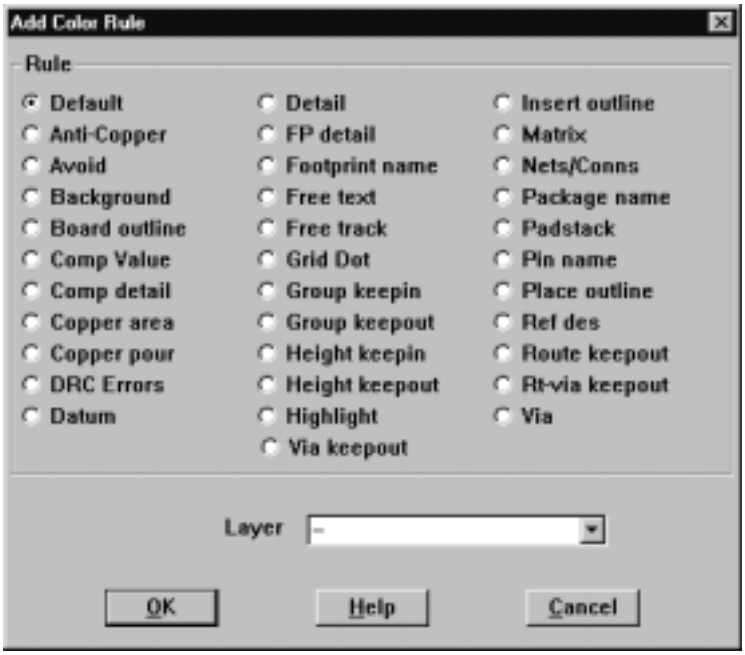
1. In the design window, choose the Color icon.

2 In the Color spreadsheet, select the layer or the object whose visibility you want to change.

3 From the Query window, choose the Invisible command. The change in visibility is apparent in the design window when you dismiss the Color spreadsheet.

To add an object to the Color spreadsheet

1 In the Color spreadsheet, choose New from the pop-up menu. The Add Color Rule dialog box appears.



2 Select the item that you want to add and specify the layer that the item is on in the Layer text box. A dash indicates “any layer,” signifying any occurrence of the object.

3 Choose the OK button.

To delete an object or layer from the Color spreadsheet

1 Select the object or layer in the Color spreadsheet and press the Delete key. The object or layer no longer appears in the design window.

Note: This procedure deletes the object or layer only from the Color spreadsheet. It does not delete the object or layer from the board.

Related topics

Assigning a color to a board layer

Assigning a color to an object

Color dialog box

Add Color Rule dialog box

Gerber plot preview

Component Selection Criteria dialog box

Using color rules

Using the Add Color Rule dialog box, you can add an object type or a layer to the Color spreadsheet. If an object type or layer is not listed in the Color spreadsheet, it will be invisible. Furthermore, the color and visibility of object types or layers not listed in the spreadsheet cannot be edited.

You can add three categories to the Color spreadsheet: a default color for a layer, a color for an object type, or a color for a specific object type on a specific layer.

If you wish to edit the visibility or color of an object on a particular layer, you must select the appropriate option and specify a layer in the edit box.

With this procedure, it is possible to assign different colors to a single object. In case of a conflict like this, the objects assigned by layer have precedence over other assignments, and the layer color has the lowest priority.

To add a category to the Color spreadsheet

1. From the Options menu, choose the Color. The Color spreadsheet appears.
2. Click the right mouse button and from the Query window, choose New. The Add Color Rule dialog box appears.
3. Select the appropriate rule and layer, and then choose OK. Layout adds the item to the Color spreadsheet.

To edit an item in the Color spreadsheet

1. From the Options menu, choose the Color. The Color spreadsheet appears.
2. Select an item in the spreadsheet.
3. Click the right mouse button and from the pop-up menu, choose a new color. You can choose a custom color by first choosing the Properties command from the pop-up menu, selecting the color from the Color dialog box, and then choosing the color from the pop-up menu.

To delete an item from the Color spreadsheet

1. From the Options menu, choose the Color. The Color spreadsheet appears.
2. Select an item in the spreadsheet.
3. Click the right mouse button and from the pop-up menu, choose Delete.

Related topics

Database Spreadsheets command (View menu)

Color spreadsheet

Layer 0 (Zero)

Layer "0" (zero) is a special layer in Layout. Any time you designate an object as being on layer "0," it will appear on all layers. The layer with the name Conn is equivalent to layer 0.

A common use for layer 0 is the Board Outline. In addition, connections, or ratsnests, are on layer 0. In the design window, you can press "0" from the keyboard to cause all unrouted connections present on the board to be redrawn to the screen. The "0" key paints the connections to the screen, and you must use the Erase Screen command of the View menu to remove them.

In the Reconn Enabled environment (using the Component Tool), the "0" layer does three things:

- If you have nothing selected and the connections are currently invisible, the "0" key will make all of the enabled connections on the board visible.
- If you have nothing selected and the connections are visible, it will make them invisible.
- If you have selected a component and you enter "0," you will toggle between two states. The default state is to show all of the connections belonging to all of the nets that connect to the component. Typing "0" from this state will toggle to a state where only the connections directly connected to the component you have selected will be visible.

Related topics

Layer number shortcuts

Zooming

To zoom in

1. From the View menu, choose the Zoom In command (View menu), or select the Zoom In button on the toolbar. The pointer changes to a Z.
2. Place pointer in one corner of the area you would like to magnify.
3. Click and drag the pointer to the opposite corner of the area you would like to magnify.
4. Press ESC to get out of zoom mode.

To zoom out

1. From the View menu, choose the Zoom Out command (View menu), or select the Zoom Out button on the toolbar.

   Zoom Out decreases the scale of the board by 2:1 each time you click the left mouse button.
2. Press ESC to get out of Zoom mode.

To Zoom Center

1. From the View menu, choose the Zoom Center command (View menu).
2. Click the mouse at the point in the design which you would like to be centered on the screen.

To Zoom with the DRC/Route Box

1. From the View menu, choose Zoom DRC/Route Box command (View menu). The pointer changes to the zoom cursor.
2. Move the pointer to any point on the screen and then click and immediately release the left mouse button. If you

press and hold the left mouse button, Layout assumes that you wish to resize the DRC Box.

You can move the DRC Box to any location within the viewing screen.

3 When the DRC Box is at the correct location, click the left mouse button again and Layout centers that location on the screen and resets the graphics so that you can view the entire routing window with as little extraneous data as possible.

4 Press ESC to get out of zoom mode.

Related topics

Zoom DRC/Route Box command (View menu)

Zoom Center command (View menu)

Zoom Out command (View menu)

Zoom In command (View menu)

DRC/Route Box command (Autoroute)

DRC/Route Box command (Fanout)

DRC/Route Box command (Delete Violating Tracks)

DRC/Route Box command (Unroute)

Design Rule Check command

Numerical boundaries for Layout

The following table identifies the upper and lower boundaries of Layout with regard to certain aspects of your board design.

Layers	
Total Layers	30
Simultaneous routing layers	16

Padstacks and Vias	
Padstacks	1000
Via Types (of the total 1000 padstack types)	
in native Layout files	16
in files translated from other EDA tools	250

Components, Footprints and Packages	
Components	32000
Minimum component rotation	1/60 of a degree
Footprints	16000
Pads per footprint	3200
Packages	5000
Gates per package	100

Nets, Connections and Obstacles	
Nets per board	10000
Connections per board	32000
Connections per net	16000
Characters per net name	100

Obstacles per board	32000
Corners per obstacle (corners per connection)	1000

Text	
Text strings	32000
Characters per text string	100

Post processing	
Post processing artwork layers	1000
Gerber apertures	2000
Drill chart symbols	40
Color assignments for screen display	100
Color assignments per artwork layer	64

Other	
Characters per reference designation	100
Error marker limit	8000

Board size and resolution			
Workspace setting	Very Small	Typical	Very Large
Imperial Board Size	17 x 17 inch	70 x 70 inch	280 x 280 inch
Imperial Resolution	1 / 960 mil	1 / 480 mil	1 / 60 mil
Metric Board Size	800 X 800 mm	1600 x 1600 mm	16,000 x 16,000 mm
Metric Resolution	1 / 100000 mm	1 / 10000 mm	1 / 1000 mm

Shortcuts

In Layout, local accelerators or shortcuts provide a quick way to perform an operation. The action of a specific shortcut key may vary depending upon what window is active and what tool is selected.

Related topics

Global action shortcuts

File menu shortcuts

Edit menu shortcuts

View menu shortcuts

Tool menu shortcuts

Options menu shortcuts

Auto menu shortcuts

Window menu shortcuts

Window and spreadsheet shortcuts

Layer number shortcuts

Global action shortcuts

Shortcuts

CTRL+C	Copy
CTRL+F or TAB	Find
CTRL+G	System settings
CTRL+O	Open file
CTRL+P	Print

CTRL+C	Copy
R	Rotate
CTRL+S	Save
CTRL+X	Delete
U	Undo
F1	Help Contents
F2	Context-sensitive help
F3	Component Selection Criteria dialog box
F5	Redraw
SHIFT+F4	Tile Windows
SHIFT+F5	Cascade Windows
i	Zoom in
O	Zoom out
PAGE UP	Move up one screen
PAGE DOWN	Move down one screen
SHIFT+PAGE UP	Move right one screen
SHIFT+PAGE DOWN	Move left one screen
HOME or F5	Redraw screen
SHIFT+HOME	Zoom All (Fit)
BACKSPACE	Erase Screen
SPACEBAR	Same as left mouse click
ENTER	Same as double mouse click
PLUS	Same as right mouse click
ESC	Same as middle mouse click
ARROW KEYS	Move pointer one grid up, left, right, or down

File menu shortcuts

New	CTRL + N
Open	CTRL + O
Save	CTRL + S
Print/Plot	CTRL + P
Library Manager	CTRL + I

Edit menu shortcuts

Undo	U
Find/Goto	CTRL + F; TAB
Select Any	ALT + S
Select Next	N
Properties	CTRL + E

View menu shortcuts

High Contrast	. (period)
Clear Screen	Backspace
Redraw	Home; F5
Query Window	Q
Zoom All (Fit)	SHIFT + HOME
Zoom Center	C
Zoom In	I
Zoom Out	O

Zoom DRC/Route Box	B
Visible <> Invisible	- (hyphen)

Tool menu shortcuts

Layer	Properties	CTRL + E
Cluster	Make	K
	Break	CTRL + K
Group	Opposite	T
	Rotate	R
	Lock	L
	Properties	CTRL + E
	Delete	CTRL + X
Matrix	Delete	CTRL + X
Component	Adjust	CTRL + J
	Opposite	T
	Rotate	R
	Lock	L
	Shove	J
	Swap	CTRL + W
	Properties	CTRL + E
	Delete	CTRL + X
Package	Properties	CTRL + E
Gate	Swap	CTRL + W
Footprint	Properties	CTRL + E
	Delete	CTRL + X
Padstack	Properties	CTRL + E
	Delete	CTRL + X

Pin	Copy	CTRL + C
	Rotate	R
	Swap	CTRL + W
	Properties	CTRL + E
	Delete	CTRL + X
Aperture	Properties	CTRL + E
	Delete	CTRL + X
Net	Unlock	CTRL + L
	Lock	L
	Change Width	W
	Properties	CTRL + E
	Delete	CTRL + X
Connection	Tack	CTRL + T
Track	Mirror	CTRL + M
	Rotate	R
	Finish	F
	Lock	L
	Unlock	CTRL + L
	Unroute	D
Track Segment	Exchange ends	X
	Change Width	W
	Delete	CTRL + X
	Begin new segment off of current segment	ALT + click
Jumper	Properties	CTRL + E
	Delete	CTRL + X
Via	Add Via	V
	Add Free Via	E

	Properties	CTRL + E
	Delete	CTRL + X
Test Point	Add Test Point	P
	Properties	CTRL + E
	Delete	CTRL + X
Text	Mirror	CTRL + M
	Rotate	R
	Properties	CTRL + E
	Delete	CTRL + X
Dimension	Properties	CTRL + E
	Delete	CTRL + X
Obstacle	Mirror	CTRL + M
	Rotate	R
	Exchange ends	X
	Finish	F
	Properties	CTRL + E
	Delete	CTRL + X
Error	Delete	CTRL + X

Options menu shortcuts

System Settings	CTRL + G

Auto menu shortcuts

Place	Component(s)	CTRL + Q

Window menu shortcuts

Cascade	SHIFT + F5
Tile	SHIFT + F4

Window and spreadsheet shortcuts

Shortcuts

SHIFT+A	Open the Place Pass spreadsheet
SHIFT+C	Open the Components spreadsheet
SHIFT+D	Open The design window
SHIFT+E	Open the Route Pass spreadsheet
SHIFT+F	Open the Footprints spreadsheet
SHIFT+G	Open the Route Spacing spreadsheet
SHIFT+H	Open the Density Graph window (Layout Plus only)
SHIFT+I	Open the Statistics spreadsheet (Layout Plus only)
SHIFT+K	Open the Packages spreadsheets
SHIFT+L	Open the Color spreadsheet
SHIFT+M	Open the Error Markers spreadsheet
SHIFT+N	Open the Nets spreadsheet
SHIFT+O	Open the Obstacles spreadsheet
SHIFT+P	Open the Apertures spreadsheet
SHIFT+Q	Open the Query window
SHIFT+R	Open the Drills spreadsheet
SHIFT+S	Open the Post Process spreadsheet
SHIFT+T	Open the Padstacks spreadsheet
SHIFT+U	Open the Route Layer spreadsheet

SHIFT+W	Open the Route Sweep spreadsheet
SHIFT+X	Open the Text spreadsheet
SHIFT+Y	Open the Layers spreadsheet

Layer number shortcuts

Layout supports up to 30 routing layers plus Layer "0" to signify all layers.

Shortcuts

0	Global layers (all layers)
1	Top
2	Bottom
3	GND
4	POWER
5	INNER1
6	INNER2
7	INNER3
8	INNER4
9	INNER5
CTRL+0	INNER6
CTRL+1	INNER7
CTRL+2	INNER8
CTRL+3	INNER9
CTRL+4	INNER10
CTRL+5	INNER11
CTRL+6	INNER12
CTRL+7	SOLDER MASK TOP
CTRL+8	SOLDER MASK BOTTOM

CTRL+9	SOLDER PASTE TOP
SHIFT+0	SOLDER PASTE BOTTOM
SHIFT+1	SILKSCREEN TOP
SHIFT+2	SILKSCREEN BOTTOM
SHIFT+3	ASSEMBLY DRAWING TOP
SHIFT+4	ASSEMBLY DRAWING BOTTOM
SHIFT+5	DRILL DRAWING
SHIFT+6	DRILL TAPE
SHIFT+7	FABRICATION DRAWING
SHIFT+9	NOTES

Layout files and file translation

4

To use Layout most effectively, it helps to be acquainted with the files provided and created by Layout. Layout looks for files in the directories that were designated as their default directories during installation.

For descriptions of the reports created by Layout, see Generate Reports dialog box.

The file LAYOUT.LOG keeps track of all session-related activity in Layout. New session information is appended to previous session information, so you may want to delete this file occasionally, since it becomes very large over time.

System files

There are many files that are installed by default when you install Layout. It can be important to understand the how each file is used by Layout.

In addition to the files listed below, Layout also provides other design files, such as Technology templates, Placement strategy files and Routing strategy files in the \TOOLS\LAYOUT\DATA directory.

These files are provided when you install Layout:

LSESSION.INI

Installed in the \TOOLS\LAYOUT directory, this file configures the session frame. Specifically, this file sets the directory paths for the various files Layout uses.

LAYOUT.INI

Contains vital information about Layout and your board. During installation, LAYOUT.INI is placed in the Layout directory. In addition, a copy of LAYOUT.INI must be located in the Capture directory in order for Capture to generate a netlist or perform forward annotation to Layout. Edit LAYOUT.INI when adding new properties, so that the properties can be passed in the netlist. If you edit the LAYOUT.INI in the Layout directory, be sure to copy the updated LAYOUT.INI to the Capture directory. LAYOUT.INI includes setup information for the following areas:

- The list of currently available libraries
- Properties that are passed from Capture to Layout
- Post processing
- Custom reports
- Default net colors by weight (priority for routing)

The LAYOUT.INI file stores settings that govern the way Layout works. It is recommended that you edit this file only if necessary.

The Layout installation places the LAYOUT.INI in the LAYOUT directory, first renaming any existing LAYOUT.INI in the directory. If you have customized the file, you are prompted to copy the changes from the original, renamed file into the newly-installed file.

The Layout installation also places a copy, with the name LAYOUT.IN~, in the DATA directory to serve as a backup in case something happens to your LAYOUT.INI file.

Layout searches for LAYOUT.INI in the following directories, in this order:

- The current design directory
- The DATA directory just below the directory that holds the executable files
- The directory that holds the Layout executable files
- The Windows directory

If you maintain a copy of LAYOUT.INI in each of several design directories, you can have distinct setups for each of your designs. If you maintain several versions of the file, take care that you edit the appropriate file for your purposes.

With version 7.0 or later of Capture and Layout, the Layout netlist utility, DSN2MNL, uses the Windows Registry and other search methods to locate LAYOUT.INI.

Layout looks for the LAYOUT.INI file in the following locations:

- The Windows registry.
- EDA_ROOT\LIBRARY directory. Used for legacy schematic Capture systems.
- EDA_ROOT\FP_LIB directory. Used for legacy schematic Capture systems.
- Current directory. Allows you to have a custom version with a design.
- Layout directory (where LAYOUT.EXE resides). This can be on the network.
- If LAYOUT.INI is not found, Layout uses the Windows search engine to search:
- The Layout directory (where LAYOUT.EXE resides).
- The current directory.
- The SYSTEM directory (Windows 95).
- The SYSTEM32, then the SYSTEM directory for Windows NT.

- The Windows directory.
- The Path environment.
- EDA_ROOT (a legacy environment variable).

LAYOUT.CTL

Installed in the \TOOLS\LAYOUT\DATA directory, this file is a control file that contains all the text strings that Layout uses. This file is a binary file, and should not be modified.

SYSTEM.PRT

SYSTEM.PRT is an ASCII file that contains information regarding the correspondence between part names and footprints. This file acts as a backup to map parts to footprints if the footprints are not defined in the schematic.

Do not modify SYSTEM.PRT. Layout provides a customizable version of this file named USER.PRT.

The SYSTEM.PRT file contains information on the correspondence between part names (the *electrical* part descriptions, such as 74LS00 or .01UF) and footprints (the *physical* part descriptions, such as DIP14 or CA300/100). Do not add parts to SYSTEM.PRT, since it is overwritten in subsequent Layout releases.

In order for AutoECO to work correctly with a non-OrCAD schematic, each part in an OrCAD netlist file (a file with an .MNL extension) must also be present in the SYSTEM.PRT file.

The format of SYSTEM.PRT is as follows:

```
!
! SYSTEM.PRT -- System Part Library
!
!
0       - DIP14,,DIP14_SO
5400    - DIP14,,DIP14_SO
7400    - DIP14,,DIP14_SO
```

An exclamation point (!) is used for a comment line.

The first column is used for the part name. When you create a schematic or parts list, you should use the part name to identify each component, so that AutoECO will be able to find the appropriate footprint(s) for each part.

The second column (filled with a "-" in the example above) is used for an optional list of packages (the part's electrical information for gate and pin swap). This information normally comes directly from the schematic, and so is not filled in here.

The third column (filled with DIP14 in the example above) is used for the default footprint that will be called up in Layout when the corresponding part name is specified.

The fourth column (filled with a blank in the example above) is used for the mirror footprint, if one exists. This is created by Layout when you mirror the part, and so is not filled in here.

Each subsequent odd-numbered column is used for any alternate footprints that you would like to have available when the part is called up. In the example above, the column is filled with DIP14_SO.

USER.PRT

USER.PRT is a copy of the SYSTEM.PRT file that you can customize, and it is located in the LAYOUT\DATA directory. USER.PRT is automatically updated during the AutoECO process. Each time an electrical part description is encountered and AutoECO is unable to match it to a footprint, AutoECO prompts you to enter a footprint name. Once the footprint is matched, AutoECO enters the reference into USER.PRT. Layout looks first in USER.PRT, then in SYSTEM.PRT to resolve part descriptions.

FMT files

Installed in the \TOOLS\LAYOUT\DATA directory, these files are used to set up the format of Gerber and other post processing output files.

LLB files

Installed in the \TOOLS\LAYOUT\LIBRARY directory, these files are Layout footprint libraries that contain the component templates that are used to design a printed circuit board (PCB).

LIS files

These files are output error listings and activity lists. In the \TOOLS\LAYOUT\LIBRARY directory, the .LIS files list the footprints in each library.

MAX files

These files are binary Layout PCBs.

MNL files

These files are binary Layout netlist files created by the Import netlist utilities and OrCAD Capture, and are used by AutoECO to create or modify PCBs.

PPF files

Installed in the \TOOLS\LAYOUT\DATA directory, these files are post processing setup files.

Related topics

Design files

MAX ASCII files

Translating other file formats into Layout files

Translating Layout files into other file formats

DXF import and export

Updating boards and libraries to Release 9 format

Converting pre-Layout v7.10 split planes

Design files

Related topics

System files

MAX ASCII files

Translating other file formats into Layout files

Translating Layout files into other file formats

DXF import and export

Updating boards and libraries to Release 9 format

Converting pre-Layout v7.10 split planes

Library files

Library files are Layout footprint libraries that contain the component templates used to design a board. Layout provides over 3000 footprints in its libraries. You may also create new footprints and custom libraries. Library files are located in the LIBRARY directory, and have .LLB extensions.

Report files

Layout generates two files that report Layout session information.

.LOG

LAYOUT.LOG, also called the session log, keeps track of all session-related activity while in Layout. New session information is appended to previous session information, so

you may want to delete LAYOUT.LOG occasionally, because the file becomes very large over time.

.LIS

.LIS files are output error listings and activity lists. In the LIBRARY directory, these files list the footprints in each library.

Netlist files

Netlist files (.MNL) are used by AutoECO to create or modify boards. Layout netlists are usually created by using the Layout tab in Capture. This causes DSN2MNL.DLL to run, creating an MNL file. You can also create an ASCII netlist file in Futurenet, OrCAD PCB II, or MAX ASCII format.

With the netlist formats below, if any net names include non-alphanumeric ASCII characters or spaces, you should enclose the entire net name in double quotes.

Futurenet

From the File menu of the Layout Session frame, select Import, then choose Futurenet Netlist. This converts a Futurenet netlist file (which may have been output by a system other than Futurenet) to MNL format.

OrCAD PCB II

From the File menu of the Layout Session frame, select Import then choose PCB II Netlist. The translator reads both the Part Name and the Footprint Name from the PCB II netlist configuration.

MAX ASCII

From the File menu of the Layout Session frame, select Import, then choose the MAX ASCII to MNL command. In order to create an MNL file, you need to have the file

SYSTEM.PRT in the DATA directory and it needs to reflect the requirements of your design.

To create an ASCII netlist file to use as a model, select one of the Layout sample netlist files then select the Export command of the File menu and choose the MNL to MAX ASCII command.

Board files

Files with a .MAX extension are Layout board files. The AutoECO process creates a .MAX file by combining the schematic netlist (.MNL) and the board or technology template (.TCH) you specify when you create a new board.

Board templates

Board templates have a .TPL extension, and consist of a board outline and basic design rules, thus acting as a foundation upon which to build a board. When starting a new board, Layout asks you to load a board template as the first step in board creation. Layout's board templates offer approximately 70 board outlines. The board outlines use the same design rules as Layout's technology template DEFAULT.TCH, which is described in Technology templates on page 127.

Note: You can create custom board templates. See Creating custom technology templates on page 226.

Technology templates

Technology templates have a .TCH extension, and enable you to set design standards for your boards quickly and easily. It may be easiest to think of a technology template as a board without physical objects or net information.

Technology templates can contain anything that can be defined and included in a board, except a netlist. At the highest level, technology templates specify the manufacturing complexity of the board, and set up a rule for the component

type used most predominantly on the board. In particular, technology templates can define the board layer structure, default grids, spacing, track widths, padstack descriptions, default colors, and can also include Gerber output settings.

Some objects on the board must be flagged as not in the netlist in a technology template, or they will be deleted during AutoECO process. These include tooling holes or mounting holes, stiffeners, mechanical parts, and any other parts on the board that are not defined in the schematic.

When you load a technology template, it replaces certain settings in the board, and ignores others. It replaces the following information:

- Placement strategy
- Routing strategy
- Number of defined layers, layer names, layer properties (such as spacing)
- Grids
- Padstacks

The following information is ignored when you load a technology template:

- Colors
- Packages
- Symbols
- Components
- Nets
- Connections
- Obstacles
- Text
- Everything else

When you use a technology template, you establish the level of manufacturing complexity your board requires. There are

three levels of manufacturing technology defined (per IPC-D-275). They provide three levels of setup, placement, and routing rules that reflect increased sophistication of tooling, materials, or processing.

- **Level A (general design complexity; preferred manufacturing)** This technology allows one track between standard DIP IC pins. Route spacing is 12 mils.
- **Level B (moderate design complexity; standard manufacturing)** This technology allows two tracks between standard DIP IC pins. Route spacing is 8 mils.
- **Level C (high design complexity; reduced ease of manufacturing)** This technology allows three tracks between standard DIP IC pins. Route spacing is 6 mils.

The technology templates included with Layout are described next.

1BET_ANY.TCH

Based on Level A as described above, a standard DIP IC pin has 62-mil pads and 38-mil drills. Routing and via grids are 25 mils, the placement grid is 100 mils, and route spacing is 12 mils.

2BET_SMT.TCH

Based on Level B as described above, it is used for surface-mount or mixed-technology boards. A standard DIP IC pin has 54-mil pads and 34 mil-drills. Routing and via grids are 81/3 mils, the placement grid is 50 mils, and route spacing is 8 mils.

2BET_THR.TCH

Based on Level B as described above, it is used for through-hole boards. A standard DIP IC pin has 54-mil pads and 34-mil drills. Routing and via grids are 20 mils, the placement grid is 100 mils, and route spacing is 8 mils.

386LIB.TCH

Used to translate files from OrCAD PCB386+.

3BET_ANY.TCH

Based on Level C as described above, a standard DIP IC pin has 50-mil pads and 34-mil drills. Routing and via grids are $12^{1}/2$ mils, the placement grid is 50 mils, and route spacing is 6 mils.

CADSTAR.TCH

Used to translate files from CadStar.

CERAMIC.TCH

Used to set up ceramic chip modules.

DEFAULT.TCH

Default technology template for typical boards. Based on Level A as described above, a standard DIP IC pin has 62-mil pads and 38-mil drills. Routing and via grids are 25 mils, the placement grid is 100 mils, and route spacing is 12 mils.

HYBRID.TCH

Used for hybrid chips.

JUMP5535.TCH

Used for single-layer boards with 55-mil vias and 35-mil drills.

JUMP6035.TCH

Used for single-layer boards with 60-mil vias and 35-mil drills.

JUMP6238.TCH

Used for single-layer boards with 62-mil vias and 38-mil drills.

MCM.TCH

Used for setting up multichip modules.

METRIC.TCH

Used for metric boards. If you are designing a board that is using metric units, you should start with the METRIC.TCH technology template to achieve the best precision.

PADS.TCH

Used to translate files from PADS.

PCAD.TCH

Used to translate files from P-CAD.

PROTEL.TCH

Used to translate files from Protel.

TANGO.TCH

Used to translate files from Tango.

TUTOR.TCH

Used with Layout’s online tutorial.

Strategy files

There are two types of strategy files in Layout: placement strategy files and routing strategy files. Although both types of files have a .SF extension, placement strategy files begin with the letters “PL.”

Placement strategy files (used for autoplacement) determine the placement of components based on different priorities, such as whether clusters are used, whether gates and pins are to be swapped, or whether you want the fastest placement.

Routing strategy files (used for autorouting) determine which default routing layers to use, when to use vias, which direction tracks should travel, which colors to use for tracks, and the size of the active routing window.

Predefined strategy files are supplied with Layout. The files are optimized for specific types of boards based on the type of components on the board, the number of layers enabled for routing, and the preferred track direction on the top layer. When creating your own strategy file, it is easiest to start by modifying one of the existing files.

If you attempt to load two strategy files, the prior strategy file is overwritten by the new one. For example, if you load a placement strategy file, and then at routing time load a routing strategy file, the routing strategy file is the one in use by Layout.

Placement strategy files

The placement strategy files provided with Layout are:

- **PLBEST.SF** - Used for the best quality placement on most boards. The completion time is generally comparable to that of PLSTD.SF.
- **PLCLUST.SF** - Used for the automatic creation of clusters, to aid in interactive placement. This strategy file can be used to advantage when interactively placing a board, especially when there is no schematic to clearly show the relationship between components. Clustered

components are represented in Layout by circles that encompass the approximate area required by the cluster.

- **PLFAST.SF** - Used to quickly complete component placement for simple boards, but it may not result in the best possible placement for complex, heavily-bussed boards. It can be used to identify potential problem spots in larger placements, to allow you to decide whether to use different-sized equivalent parts, or to determine which side of the board to use to place surface-mount parts.
- **PLFINISH.SF** - Begins where PLCLUST.SF leaves off and finishes placement using PLBEST.SF.
- **PLSTD.SF** - Used for a high-quality placement which, in most cases, comes close to optimum. Because of its precision, this strategy takes substantially longer to complete placement than other strategies. This strategy does not include gate and pin swap.

Note: When you translate a PCB 386+ library file into Layout, Layout prompts you to use 386LIB.TCH, as well as 386LIB.SF. If you accept this suggestion, Layout maintains the color setup of your original PCB 386+ library.

Routing strategy files

The routing strategy files provided with Layout are listed below. Note that the number of board layers given indicates the number of *routing* layers (not total layers) on a board.

STD.SF is the standard strategy file that is automatically loaded into each board as it is translated into Layout's binary format. All other strategies are derived from this one. It exists as a separate file in the DATA directory and must be present in the directory in order to translate a board into Layout. You can also load this strategy file and use it with boards that were not translated. In addition, loading STD.SF is a way to make all objects visible, which is helpful when troubleshooting.

- A 2, 4, 6, or 8 indicates the number of *routing* layers (not total layers) on a board.

- An H indicates a horizontal primary routing direction on layer one.
- A V indicates a vertical primary routing direction on layer one.
- THR is for through-hole boards.
- SMD is for two-layer, single-sided, or double-sided, surface-mount or mixed-technology boards.
- SM1 is for single-sided, surface-mount boards. Use these strategy files for multilayer surface-mount or mixed-technology boards with active components on the component side only.
- SM2 is for double-sided, surface-mount boards. Use these strategy files for multilayer surface-mount or mixed-technology boards with active components on the component and solder sides.

Note: The strategy files included with Layout have been optimized to route typical surface-mount or through-hole boards of two to eight routing layers. For boards with more than eight routing layers, you should modify an eight-layer strategy file, keeping the same pattern.

2__SMD_H.SF

Used for a two-layer, single-sided or double-sided, surface-mount or mixed-technology board, with layer one horizontal.

2__SMD_V.SF

Used for a two-layer, single-sided or double-sided, surface-mount or mixed-technology board, with layer one vertical.

2__THR_H.SF

Used for a two-layer, through-hole board, with layer one horizontal.

2__THR_V.SF

Used for a two-layer, through-hole board, with layer one vertical.

386LIB.SF

Used for libraries translated from OrCAD PCB386+.

4__SM1_H.SF

Used for a four-layer, single-sided, surface-mount or mixed-technology board, with layer one horizontal.

4__SM1_V.SF

Used for a four-layer, single-sided, surface-mount or mixed-technology board, with layer one vertical.

4__SM2_H.SF

Used for a four-layer, double-sided, surface-mount or mixed-technology board, with layer one horizontal.

4__SM2_V.SF

Used for a four-layer, double-sided, surface-mount or mixed-technology board, with layer one vertical.

4__THR_H.SF

Used for a four-layer, through-hole board, with layer one horizontal.

4__THR_V.SF

Used for a four-layer, through-hole board, with layer one vertical.

6__SM1_H.SF

Used for a six-layer, single-sided, surface-mount or mixed-technology board, with layer one horizontal.

6__SM1_V.SF

Used for a six-layer, single-sided, surface-mount or mixed-technology board, with layer one vertical.

6__SM2_H.SF

Used for a six-layer, double-sided, surface-mount or mixed-technology board, with layer one horizontal.

6__SM2_V.SF

Used for a six-layer, double-sided, surface-mount or mixed-technology board, with layer one vertical.

6__THR_H.SF

Used for a six-layer, through-hole board, with layer one horizontal.

6__THR_V.SF

Used for a six-layer, through-hole board, with layer one vertical.

8__SM1_H.SF

Used for an eight-layer, single-sided, surface-mount or mixed-technology board, with layer one horizontal.

8__SM1_V.SF

Used for an eight-layer, single-sided, surface-mount or mixed-technology board, with layer one vertical.

8__SM2_H.SF

Used for an eight-layer, double-sided, surface-mount or mixed-technology board, with layer one horizontal.

8__SM2_V.SF

Used for an eight-layer, double-sided, surface-mount or mixed-technology board, with layer one vertical.

8__THR_H.SF

Used for an eight-layer, through-hole board, with layer one horizontal.

8__THR_V.SF

Used for an eight-layer, through-hole board, with layer one vertical.

FAST_H.SF

Used for quickly checking on a particular placement, with layer one horizontal.

FAST_V.SF

Used for quickly checking on a particular placement, with layer one vertical.

JUMPER_H.SF

Used for boards with jumper layers, with layer one horizontal.

JUMPER_V.SF

Used for boards with jumper layers, with layer one vertical.

REROUT_H.SF

Used for rerouting boards, with layer one horizontal.

REROUT_V.SF

Used for rerouting boards, with layer one vertical.

STD.SF

Used for the default routing strategy. It is automatically loaded into each board if the board is translated into Layout's binary format. You can also use this strategy file with boards that are not translated.

VIARED_H.SF

Used for a via-reduce sweep on a completely routed board, with layer one horizontal.

VIARED_V.SF

Used for a via-reduce sweep on a completely routed board, with layer one vertical.

Note: For information on modifying strategy files, see Chapter 3, "*Using routing strategy files*" in the *OrCAD Layout Autorouter User's Guide*.

MAX ASCII files

You can use an ASCII file to edit your board, or create a design from a schematic capture application that supports ASCII output. After you create an ASCII file, you can edit the file in any text editor before reading it into Layout.

This command is most often used to create a parts list manually.

If you have a schematic capture application that creates an ASCII file in the Layout format, you can import the file into Layout to create a new design. You can also create the ASCII netlist using a text editor.

Related topics

System files

Design files

Translating other file formats into Layout files

Translating Layout files into other file formats

DXF import and export

Updating boards and libraries to Release 9 format

Converting pre-Layout v7.10 split planes

Exporting and importing ASCII files

Since a Layout netlist file (with an .MNL extension) is in binary format, you have to create a text (ASCII) version of the netlist file in order to view it. Once you create the ASCII file, you can open it in a text editor such as Notepad.

To export an .MNL file to ASCII

1. Open the session frame. If you have a board file open, you must close it first.
2. From the Export menu, choose MNL to ASCII. The MAX Netlist Binary Input dialog box displays.
3. Select the .MNL file you want to export to ASCII, then choose the Open button. The MAX ASCII dialog box displays.
4. Enter the filename and directory of the new ASCII file, then choose the Save button. Layout exports the .MNL file to ASCII.

To import an ASCII file to Layout

1. Open the session frame. If you have a board file open, you must close it first.
2. From the Import menu, choose MAX ASCII to MNL. The MAX ASCII dialog box displays.
3. Find and select the .ASC (ASCII) file you want to import to Layout, then choose the OK button. The MAX Netlist dialog box displays.
4. Enter the filename and directory of the new .MNL file, then choose the OK button. Layout creates the .MNL file.

Related topics

MAX ASCII file general format

Attribute data format

Component data format

Net data format

Package and Symbol data format

MAX ASCII file general format

You can edit your Layout boards while they are in ASCII format. An .MNL file is not required to create an ASCII file: you can create ASCII files from schematic capture applications that produce ASCII files, and you can edit an ASCII file using any text editor. Any red text in the format descriptions below indicate text that must be present in an ASCII file. Blue text indicates parameters that must be defined for the line of data.

General format

MAX ASCII files use the following general format:

```
*MASSTECK
*USER    user-param
*INCH    inch-param
*METRIC  metric-param
*START
File data
Package and Symbol data
Component data
Net data
*END
```

User-param

Specifies the number of base units or microns equal to one user unit. With this option, you can use coordinates of up to one micron when placing components through the ASCII interface. The default value for this parameter is 60.

Inch-param

Specifies the default granularity of the design, and is the base unit size used by the METRIC.TCH file. The default value for this parameter is 0.00001666666666666667.

Metric-param

Specifies use of metric units. If set to YES, Layout uses metric units. The default value for this parameter is NO.

File data

File data provides information about the design file using the standard attribute data format. Layout generates these lines when you export an .MNL file to ASCII. For more information, see Attribute data format.

Package and Symbol data

Package data provides pin information for each device. Symbol data matches footprint pins with pins that appear in the netlist and pins that appear in the package. For more information on the package and symbol data format, see Package and Symbol data format.

Component data

Component data provides component information. A component is specified on a single line, with optional attribute lines immediately following. For more information on the component data format, see Component data format.

Net data

Net data provides net information. A single net is specified on one or two lines, with optional attribute lines immediately following. For more information on the net data format, see Net data format.

Related topics

Exporting and importing ASCII files

Attribute data format

Component data format

Net data format

Package and Symbol data format

Attribute data format

Attribute lines may follow file data, component data, and net data. Each type of data may have one or more attribute lines, or may have none. Attribute lines have the following format:

```
*ATTR KEY=Name Type Identifier Val1 Val2 Val3 Val4
```

where:

- **Name** - specifies the attribute name.
- **Type** - specifies Count (for files), Comp (for components), or Net (for nets).
- **Identifier** - specifies count, component, or net (U1 for example with components).
- **Val1** - a special parameter reserved for Layout. Set this parameter to "Count" for component and net attributes.
- **Val2** - may be a special file parameter reserved for Layout, or any of the following parameters explained below: PartNum, PowerPin, SpacingByLayer, PlaneLayers. ViaPerNet, or WidthByLayer.
- **Val3** - may be the filename and directory, or the value of any of the following parameters explained below: FPList, PartNum, PowerPin, SpacingByLayer, PlaneLayers, ViaPerNet, and WidthByLayer.
- **Val4** - a special parameter reserved for Layout. Set this parameter to 0 for component and net attributes.

In the following parameter descriptions and examples, blue text indicates how the parameter is used.

FPList

FPList is a comma-delimited list of alternate footprints to be attached to the component, for ease of switching between footprints. Set the Val2 parameter to 1 (shown in red).

```
*ATTR Key="AltSym" "Comp" "U2" "Count" 1 "DIP14\SO"
0
```

PartNum

PartNum is the customer part number, which is generally unique for each customer, as opposed to the PartShape, which is more generic. This part number usually designates the exact part, including manufacturer and case type.

```
*ATTR Key="Named" "Comp" "U2" "Count" "PartNum"
"489746" 0
```

PowerPin

PowerPin defines non-wired pins (such as unusual voltages) as belonging to a particular net, and appears as part of the netlist. This is typically used to override the standard GND or VCC attachments to the particular pins of the IC.

SpacingByLayer

SpacingByLayer presets the net spacing criteria per layer.

```
*ATTR Key="Named" "Net" "CK2" "Count"
"SpacingByLayer" "TOP=7,BOT=12" 0
```

Plane Layers

Plane Layers presets a net to a given plane layer.

```
*ATTR Key="Named" "Net" "GND" "Count"
"ThermalLayers" "GND" 0
```

ViaPerNet

ViaPerNet attaches specific vias to specific nets, such as VCC and GND.

```
*ATTR Key="Named" "Net" "CK2" "Count" "ViaPerNet"
"VIA1" 0
```

WidthByLayer

WidthByLayer presets the net widths per layer.

```
*ATTR Key="Named" "Net" "CK2" "Count"
"WidthByLayer" "TOP=7,BOT=12" 0
```

Related topics

Exporting and importing ASCII files

MAX ASCII file general format

Component data format

Net data format

Package and Symbol data format

Component data format

There must be one or more component lines for each component in the design, and a component may have one or more optional attribute lines. In the format below, red text indicates parameters that must be specified. Blue text indicates parameters that may be specified in any order in the line, provided that they appear after the parameters described here in red text. Component lines use the following format:

```
*COMP Component PartShape CompFixed CompGroup
CompKey CompLoc CompLocked CompRot Footprint
MirrorFootPrint PackageNum
```

In the following parameter descriptions and examples, blue text indicates how the parameter is used.

CompFixed

CompFixed is used to permanently fix components, such as edge connectors, to a PCB.

```
*COMP U1 "74LS00" Fixed
```

CompGroup

CompGroup is the floor planning attribute that can be set in the schematic in order to guide both you and the autoplacement routines in placing the board correctly.

```
*COMP U2 "74LS00" [2100, 2200] Rot=270[00] Group=2
```

CompKey

CompKey is used to designate a component as the key component in a given group. The key component of a group is placed first, with all of the other components in the group placed in proximity to it.

```
*COMP U3 "74LS00" Key
```

CompLoc

CompLoc is the component location in the Layout design in "user units," which are typically either mils or microns. The user-unit-to-base-unit conversion is defined in LAYOUT.INI under [SCHGLOBAL].

```
*COMP U4 "74LS00" [1000, 1000] Rot=270[00] Group=2
```

CompLocked

CompLocked is used to temporarily lock components, such as pre-placed memory arrays, to a PCB.

```
*COMP U5 "74LS00" Locked
```

Component

Component specifies a component for the design. All the information on a component line applies to the specified component.

```
*COMP U6 "74LS32"
```

CompRot

CompRot is the component rotation (in degrees and minutes counterclockwise) from the 0 rotation as the component was defined in the library.

```
*COMP U7 "74LS00" [2100, 2200] Rot=270[00] Group=2
```

Footprint

Footprint is an explicit definition of the footprint name.

```
*COMP U8 "74LS00" Footprint="DIP14"
Package="74LS00"
```

MirrorFootPrint

MirrorFootPrint defines an explicit mirror shape for the component, in case you do not want Layout to perform mirroring.

```
*COMP U9 "74LS00" Footprint="DIP14"
MFootprint="DIP14-M"
```

PackageNum

PackageNum identifies the package information for the component.

```
*COMP U10 "74LS32" Footprint="DIP14" MFootprint=""
Package=3
```

PartShape

PartShape is the generic part number, such as 74LS04 or CKO5, that is generally understood to mean a certain part throughout the industry, but may not identify the part as to manufacturer or case type. If no footprint is defined, or the correct footprint cannot be found using the footprint name, this field will be matched against the left-hand column of the SYSTEM.PRT cross-reference list to find a footprint.

```
*COMP U11 "74LS32" Footprint="DIP14" MFootprint=""
Package=3
```

Note: Layout installs the LAYOUT.INI file in the WINDOWS directory, and it is recommended that you do not move it. Layout can find the LAYOUT.INI only if it is in one of four directories: the design directory, the DATA directory just below the directory that holds the executable files, the directory that holds the Layout executable files, or the WINDOWS directory. If you keep a copy of LAYOUT.INI in each of several design directories, you can have different configurations for the different designs.

Related topics

Exporting and importing ASCII files

MAX ASCII file general format

Attribute data format

Net data format

Package and Symbol data format

Net data format

There may be one or more net lines for nets in the design. One net may have one or two lines, with one or more optional attribute lines. In the format below, red text indicates required parameters. Blue text indicates optional parameters that can intermix in any order with other blue-text parameters. Magenta text indicates optional parameters that can intermix in any order with other magenta-text parameters. Net data uses the following format:

```
*NET NetName ConnWidth MinWidth MaxWidth Highlight
NetGroup NetLev NetWeight ReconnType TestPoint Pin1
Pin2 ...
```

or

```
*NET NetName ConnWidth MinWidth MaxWidth Highlight
NetGroup NetLev NetWeight ReconnType TestPoint
*NET NetName Pin1 Pin2 ...
```

In the following parameter descriptions and examples, blue text indicates how the parameter is used.

ConnWidth

ConnWidth sets only the "Conn" width per net, leaving MinWidth and MaxWidth at their defaults.

```
*NET "GND" W:12
```

Highlight

Highlight highlights various nets in Layout.

```
*NET "GND" Highlight
```

MaxWidth

MaxWidth designates a maximum range for the net widths.

```
*NET "GND" Max:12
```

MinWidth

MinWidth designates a minimum range for the net widths.

```
*NET "GND" Min:12
```

NetGroup

NetGroup groups various nets in Layout from the schematic. They can then be colored or selected, according to group, for editing or routing.

```
*NET "GND" Group:2
```

NetLev

NetLev specifies the layers available for routing. NetLev may be specified as a number, or as layer names. If NetLev is specified as layer names, the names must be enclosed in double quotes and separated by commas.

```
*NET "CK2" W:12 NetLev:"TOP,BOT" Group:2
Reconrule:ECL
```

or

```
*NET "CK2" W:12 NetLev:65535 Group:2 Reconrule:ECL
```

NetWeight

NetWeight attaches greater or lesser importance to each net on the PCB. This is a number from 1 to 100, and is automatically color-coded upon entry.

```
*NET "GND" Weight:60
```

ReconnType

ReconnType sets the reconnect type per net from among STD, HORZ, VERT, MIN, MAX, or ECL.

```
*NET "CK2" W:12 NetLev:"TOP,BOT" Group:2
Reconrule:ECL
```

TestPoint

TestPoint presets nets as needing a testpoint during the DFM phase of board design.

```
*NET "GND" TestPtNeeded
```

Note: Layout installs the LAYOUT.INI file in the WINDOWS directory, and it is recommended that you do not move it. Layout can find the LAYOUT.INI only if it is in one of four directories: the design directory, the DATA directory just below the directory that holds the executable files, the directory that holds the Layout executable files, or the WINDOWS directory. If you keep a copy of LAYOUT.INI in each of several design directories, you can have different configurations for the different designs.

Related topics

Exporting and importing ASCII files

MAX ASCII file general format

Attribute data format

Component data format

Package and Symbol data format

Package and Symbol data format

The package and symbol lines of the ASCII file are optional. Symbol lines are only present to match footprint pins with both the pins that appear in the netlist and the pins that appear in the package. For example, pin number 2 of the package could be matched up with pin number 4 of the footprint when the netlist is merged with the Layout design file during AutoECO. In the format below, red text identifies parameters that must be specified. The package and symbol lines use the following format:

```
*PACKAGE PackageNum PartShape
1 Device PinName GateGroup PinGroup ECLType
2 Device PinName GateGroup PinGroup ECLType
...
*SYMBOL PartShape
1 Match1
2 Match2
...
```

There must be a line in the Package section for each pin in the package. Power and ground pins are not counted.

In the following parameter descriptions and examples, blue text indicates how the parameter is used.

Device

Device identifies the device in the package.

```
1 "A" "I0" 1 0 SOURCE
```

ECLType

ECLType defines a pin as a Source, Load, or Target in terms of daisy-chain reconnection.

```
1 "A" "I0" 1 0 SOURCE
```

GateGroup

GateGroup identifies any gate swapping restrictions within a component. In order to be swapped, two gates must belong to the same gate group.

```
1 "A" "I0" 1 0 SOURCE
```

Match*N*

Match*N* (where N is an integer greater than 0) matches a footprint pin with both the pin that appears in the netlist and the pin that appears in the package.

```
1 "1"
```

PackageNum

PackageNum identifies the package the information applies to. Symbol information must immediately follow the package information for the same package.

```
*PACKAGE 1 "74LS00"
```

PartShape

PartShape is the generic part number, such as 74LS04 or CKO5, that is generally understood to mean a certain part throughout the industry, but may not identify the part as to manufacturer or case type. If no footprint is defined, or the correct footprint cannot be found using the footprint name, this field will be matched against the left-hand column of the SYSTEM.PRT cross-reference list to find a footprint.

```
*PACKAGE 1 "74LS00"
```

PinGroup

PinGroup identifies any pin swapping restrictions within a gate. In order to be swapped, two pins must belong to the same gate, and to the same pin group.

```
1 "A" "I0" 1 0 SOURCE
```

PinName

PinName identifies the pin on the device that the attributes apply to (for the line of data).

```
1 "A" "I0" 1 0 SOURCE
```

Related topics

Exporting and importing ASCII files

MAX ASCII file general format

Attribute data format

Component data format

Net data format

Package and Symbol data format

Translating other file formats into Layout files

Related topics

System files

Design files

MAX ASCII files

Translating Layout files into other file formats

DXF import and export

Updating boards and libraries to Release 9 format

Converting pre-Layout v7.10 split planes

PADS Import

The MAXPADX.EXE file translates a PADS-PCB or a PADS-2000 file with an .ASC extension into an OrCAD binary file with a .MAX extension.

You cannot import a board created in one application, such as PADS, place and route it using Layout, then export it to an application other than PADS---the import application must be the same as the export application.

PADS Input/Output Files

Input files

There are three types of input files used by this translator:

- **Input PADS ASCII File -** The input file from PADS should be output using the ASCII out utility.
- **PADS INI File -** The supplied file PADS.INI holds the layer mapping and options/defaults information that the ASC to MAX translator uses to equate PADS layers with Layout layers. You can also create your own custom INI file by copying PADS.INI and using the copy here. When you edit this information using the dialog, it gets written to the selected INI file for persistence.
- **Layout Technology/Template File -** The supplied file PADS.TCH is an empty Layout database that contains the default Layout layer stackup, post processing defaults, and 16 empty via padstacks. You may create your own custom template file if you prefer, for instance, one containing a C sized sheet format.

Output files

The output file is a binary file in Layout's MAX format.

If you select "Overwrite existing file", the translator will allow you to overwrite an existing MAX file by the same name.

You can use the Browse buttons to find the files you need.

PADS Layer Mapping

The layer mapping tab allows you to easily map your PADS layers to the corresponding Layout layers.

The default layer mapping for most common PADS layer setups is provided in PADS.INI, which is read into the layer mapping dialog for easy editing. Changes are saved to PADS.INI or the INI file of your choice when you select the Apply button or press OK.

Each PADS layer is listed once in the left hand listbox, and each Layout layer is listed in the right hand listbox. For each PADS layer, you can select one or more Layout layers to map the information to upon translation. For instance, you would typically map the information on the Layer 1 to Layout's TOP layer, and Layer 4 to Layout's BOTTOM layer.

The DEFAULT layer is a special designation that means to map the layer using the default setting in PADS. For instance, the bottom layer in PADS is often 2, 4, 6, or 8, depending on the number of layers on the board. DEFAULT means map it appropriately.

You can also map more than one PADS layer to the same Layout layer.

PADS Layers

This is the list of layers which appears in the left hand column of the [PADSXMAPPING] section of PADS.INI.

Add PADS Layer

You can type in a PADS layer name here to add it to the listbox. Then press <Enter> or the "Add" button to add the name to the list.

Delete PADS Layer

You can select a layer and press the "Delete" button to remove it from the list. This will remove the name entirely from the PADS.INI file when you select Apply or OK. If you wish to keep the layer name for reference but not have it mapped to anything, you have the option of removing all mappings from the layer instead.

Layout Layers

This is a list of the standard Layout layers. It uses the "Nickname" of the layer, which is a three letter abbreviation for each of the layers. For instance, BOT is the bottom (solder) layer, SMT is the soldermask top layer, DRD is the drill drawing layer. The Layer spreadsheet in Layout lists the names and nicknames of all of the layers.

Note that you can select as many Layout layers as you wish to map from each PADS layer. If you deselect all Layout layers and press the "Edit Map" button, you remove the link between the selected PADS layer and Layout without actually removing the affected layers from the list.

Add Layout Layer

You can add your own Layout layer to the list using this edit box. Then press <Enter> or the "Add" button to add the name to the list.

Delete Layout Layer

This allows you to delete a Layout layer from the list. Note that all mappings that use that layer will be affected. If you delete

the layer and don't get a warning, it means that there were no more remaining mappings to that layer and it's safe to delete it.

Edit Map

The Edit Map button creates the current layer mapping scheme in memory. However, the mapping scheme doesn't get written to PADS.INI until you press OK or Apply. If you press Cancel, the edit will be aborted, and will not be written to PADS.INI.

To use the PADS translator

1. In the PADS software, choose "Basic units" before you generate an output file.
2. In the PADS software, choose "All" from the ASCII output selection box.
3. In the PADS software, save your file as an ASCII file with an .ASC extension.
4. From the Import menu in the session frame, choose PADS PCB. The PADS-PCB ASCII Input dialog box displays.
5. Select a file (with an .ASC extension), then choose the OK button. The MAX Board dialog box displays.
6. Enter a filename with a .MAX extension, then choose the OK button. The Technology dialog box displays.
7. Select the PADS.TCH file, then choose the OK button. The PADS-PCB Extractor dialog box displays.
8. When the message "Processing completed" displays, choose the OK button to dismiss the dialog box.

Related topics

PADS PCB command (Import)

PADS PCB command (Export)

Translating other file formats into Layout files

MAX ASCII files

Creating a new Layout project from a Capture design

P-CAD Import

P-CAD Input/Output Files

Input files

There are four types of input files used by this translator:

- **Input P-CAD PDIF File -** The input file from P-CAD should be output using the PDIF out utility, and should have apertures and padstacks included if possible.
- **P-CAD INI File -** The supplied file PCAD.INI holds the layer mapping and options/defaults information that the PDIF to MAX translator uses to equate P-CAD layers with Layout layers, and also to establish defaults for any padstack information that might be missing from the PDIF file. You can also create your own custom INI file by copying PCAD.INI and using the copy here. When you edit this information using the dialog, it gets written to the selected INI file for persistence.
- **Layout Technology/Template File -** The supplied file PCAD.TCH is an empty Layout database that contains the default Layout layer stackup, post processing defaults, and 16 empty via padstacks. You may create your own custom template file if you prefer, for instance, one containing a C sized sheet format.
- **P-CAD Drill Table File -** The *.TBL (drill table) file that goes along with your P-CAD database can be used here to input the correct drill sizes. The PDIF file does not typically carry this information. If you do not have one, just use the default PCAD.TBL file supplied.

Output files

The output file is a binary file in Layout's MAX format.

If you select "Overwrite existing file", the translator will allow you to overwrite an existing MAX file by the same name.

You can use the Browse buttons to find the files you need.

P-CAD Options/Defaults (Import)

This dialog tab allows you to set various options and defaults so that missing information from the PDIF file will be filled in with reasonable defaults. Changes are saved to PCAD.INI or the INI file of your choice when you select the Apply button or press OK.

Drill Table Default Directory

This is the default directory for drill table file search. Possible values are the Data directory or the Working directory.

If you generally use the same drill table for all of your PDIF translations, then you should place the .TBL file in the \TOOLS\LAYOUT\DATA directory and use the DATA value.

If you generally use a different drill table for each of your PDIF files, then use the PROJECT value, and the translator will look in your project directory for a .TBL file.

Use the supplied PDIF.TBL in the DATA directory if you do not have your own.

Allow Undefined Pads

Check this box if you want undefined padstacks in your PDIF file to remain that way. This would be the case if all of your padstack definitions are included, and particularly if you have the aperture table included as well. If you want undefined pads to pick up the STDPAD size, leave this unchecked.

Standard Pad Size

This is the default pad size in mils if the pad is undefined. If you choose "Allow Undefined Pads", this value will be ignored.

Otherwise, if you haven’t included padstacks or apertures in your PDIF file, you should leave "Allow Undefined Pads" unchecked, and then this value will assure that a reasonable default value is assigned to the otherwise undefined padstacks.

Plane Oversize

This is the default plane layer clearance pad oversize (diameter) in mils if plane layer pad is undefined. Used only if both TOP and BOT pads are defined, is based on TOP layer pad size.

Soldermask Oversize

This is the default soldermask pad oversize (diameter) in mils if soldermask pad is undefined. Top side soldermask is based on TOP layer pad. Bottom side soldermask is based on BOT layer pad.

Pastemask Oversize

This is the default solder paste pad oversize (diameter) in mils if solderpaste pad is undefined. Top side pastemask is based on TOP layer pad. Bottom side pastemask is based on BOT layer pad.

Component Drill Undersize

This is the default drill undersize (diameter) if drill is undefined for through hole pads. Used only if both TOP and BOT pads are defined, is based on TOP layer pad size. Note the negative sign denoting that this is the amount *under* the pad size for the drill.

Via Drill Undersize

This is the default drill undersize (diameter) for vias if drill is undefined for via pads. Vias are always assumed to have a drill. Note the negative sign denoting that this is the amount *under* the pad size for the drill.

Component Drill Minimum

This is the minimum allowable drill for component through pads. If the calculated drill using the undersize value is less than the minimum, this value will be substituted for calculated drill.

Via Drill Minimum

This is the minimum allowable drill for vias. If the calculated drill using the undersize value is less than the minimum, this value will be substituted for calculated drill.

P-CAD Layer Mapping (Import)

The layer mapping tab allows you to easily map your P-CAD layers to the corresponding Layout layers.

The default layer mapping for most common P-CAD layer setups is provided in PCAD.INI, which is read into the layer mapping dialog for easy editing. Changes are saved to PCAD.INI or the INI file of your choice when you select the Apply button or press OK.

Each P-CAD layer is listed once in the left hand listbox, and each Layout layer is listed in the right hand listbox. For each P-CAD layer, you can select one or more Layout layers to map the information to upon translation. For instance, you would typically map the information on the PADCOM layer to Layout's TOP layer, and the BARINT layer to Layout's layers IN1-I12.

You can also map more than one P-CAD layer to the same Layout layer. For instance, COMP, BARCOM, and FLCOMP all map to the TOP layer in Layout.

P-CAD Layers

This is the list of layers which appears in the left hand column of the [PCADXMAPPING] section of PCAD.INI. Other than the P-CAD system layers (which start with ‘$’) you should include any layers that contain information that you want sent over to Layout.

Add P-CAD Layer

You can type in a P-CAD layer name here to add it to the listbox. Then press <Enter> or the "Add" button to add the name to the list.

Delete P-CAD Layer

You can select a layer and press the "Delete" button to remove it from the list. This will remove the name entirely from the PCAD.INI file when you select Apply or OK. If you wish to keep the layer name for reference but not have it mapped to anything, you have the option of removing all mappings from the layer instead.

Layout Layers

This is a list of the standard Layout layers. It uses the "Nickname" of the layer, which is a three letter abbreviation for each of the layers. For instance, BOT is the bottom (solder) layer, SMT is the soldermask top layer, DRD is the drill drawing layer. The Layer spreadsheet in Layout lists the names and nicknames of all of the layers.

Note that you can select as many Layout layers as you wish to map from each P-CAD layer. If you deselect all Layout layers and press the "Edit Map" button, you remove the link between the selected P-CAD layer and Layout without actually removing the affected layers from the list.

Add Layout Layer

You can add your own Layout layer to the list using this edit box. Then press <Enter> or the "Add" button to add the name to the list.

Delete Layout Layer

This allows you to delete a Layout layer from the list. Note that all mappings that use that layer will be affected. If you delete the layer and don't get a warning, it means that there were no more remaining mappings to that layer and it's safe to delete it.

Edit Map

The Edit Map button creates the current layer mapping scheme in memory. However, the mapping scheme doesn't get written to PCAD.INI until you press OK or Apply. If you press Cancel, the edit will be aborted, and will not be written to PCAD.INI.

P-CAD Layer Type

Flash (Aperture padstack) layers must be designated so that the translator will recognize these layers and give them a higher priority than graphic padstack layers.

Pad (Graphic padstack) layers must be designated as such so that the shapes defined on those layers will be correctly interpreted as padstack definitions.

Other layer types need not be otherwise identified. When mapped to a Layout layer, objects such are rectangles and circles on those layers will be interpreted as tracks or obstacles rather than pads.

P-CAD user-defined attribute translation

You can import user-defined attributes into Layout by placing an attribute key into the Layout .INI file. The procedure below uses the following user-defined attribute as an example:

(At MyUserDefinedProp AaBbCc 500.0 500.0).

To import user-defined attributes

1. Open the LAYOUT.INI file, using Notepad or other text editor.
2. Locate the [SYMATTR] section.
3. Scroll to the last line of the section and type the attribute name. In this case it would be MyUserDefinedProp.
4. Assign a starting number of 200, or any other reasonable unused number. The entry will look like this:

   MyUserDefinedProp = 200.
5. If you have Layout open, you must restart it so it reads the modified LAYOUT.INI file.

If you were importing a file that contained a component with the example user-defined attribute, Layout would place the attribute value of AaBbCc, at X,Y position 500,500 relative to the component.

Related topics

P-CAD PCB command (Import)

P-CAD PCB command (Export)

IDF Import

The IDF to Layout translator can be used to translate Intermediate Drawing Format (IDF) files into Layout. Note that IDF files come in pairs: the board file (.EMN) contains the board outline, keepouts, and component placements, while the library file (.EMP) contains additional data for each component placed in the board file, such as the component placement outline and the component height.

Importing a board using the IDF to Layout translator

1. Create a netlist with footprints and other necessary information.
2. Create a directory in which the schematic design, netlist, and board will coexist and put the schematic design (if you have it) and netlist in it. OrCAD provides a directory (\TOOLS\LAYOUT\DESIGN) for this purpose.
3. Save your board design in IDF format, using an .EMN extension for the board file and an .EMP extension for the library file, then put the .EMN and .EMP files in the same directory as your schematic design and netlist.
4. From Layout's session frame, choose File, Import, IDF to Layout. The IDF to Layout dialog box appears.
5. In the Input IDF Board File text box, supply a name for the board file (.EMN).
6. In the Input IDF Library File text box, supply a name for the library file (.EMP).
7. In the Input Technology or Layout File text box, supply a name for the technology or layout file.
8. In the Output Layout File text box, supply a name for the output .MAX file.
9. Select the following options as desired, then choose the Translate button.
10. From the Layout session frame's File menu, choose New. The Load Template File dialog box appears.
11. Change the Files of type to Board (*.MAX), locate and select the new .MAX file, then choose the Open button. The Load Netlist Source dialog box displays.
12. Select a netlist file (.MNL), then choose the Open button. The Save File As dialog box displays.
13. Specify a name for the new board, then choose the Save button. The AutoECO process begins.
14. If necessary, respond to the Link Footprint to Component dialog box (choose the dialog box's Help button for an explanation of the dialog box options).

15 AutoECO finishes, and you see the components in the design window.

Related topics

IDF to Layout dialog box

IDF to Layout command (Import)

IDF Export

Layout to IDF dialog box

Layout to IDF command (Export)

Protel Import

The MAXPROTX.EXE file translates a Protel Autotrax file or a Protel for Windows file with a .PCB extension into an OrCAD binary file with a .MAX extension.

You cannot import a board created in one application, such as Protel, place and route it using Layout, then export it to an application other than Protel---the import application must be the same as the export application.

To use the Protel translator

1 In the Protel for Windows software, save your file as a 2.8 ASCII file with a .PCB extension.

2 From the Import menu in the session frame, choose Protel PCB. The Protel PCB dialog box appears.

3 Select the file (with a .PCB extension), then choose the OK button. The MAX Board dialog box appears.

4 Enter a filename with a .MAX extension, then choose the OK button. The Technology dialog box appears.

5 Select the PROTEL.TCH file, then choose the OK button. The Protel Extractor dialog box appears.

6 When the message "Processing completed" appears, choose the OK button to dismiss the dialog box.

Protel Input/Output Files (Import)

Input files

There are three types of input files used by this translator:

- **Input Protel PCB File -** The input file from Protel is the standard 2.8 ASCII database.
- **Protel INI File -** The supplied file PROTEL.INI holds the layer mapping and information that the PCB to MAX translator uses to equate Protel layers with Layout layers. You can also create your own custom INI file by copying PROTEL.INI and using the copy here. When you edit this information using the dialog, it gets written to the selected INI file for persistence.
- **Layout Technology/Template File -** The supplied file PROTEL.TCH is an empty Layout database that contains the default Layout layer stackup, post processing defaults, and 16 empty via padstacks. You may create your own custom template file if you prefer, for instance, one containing a C sized sheet format.

Output files

The output file is a binary file in Layout's MAX format.

If you select "Overwrite existing file", the translator will allow you to overwrite an existing MAX file by the same name.

You can use the Browse buttons to find the files you need.

Protel Layer Mapping (Import)

The layer mapping tab allows you to easily map your Protel layers to the corresponding Layout layers.

The default layer mapping for most common Protel layer setups is provided in PROTEL.INI, which is read into the layer mapping dialog for easy editing. Changes are saved to PROTEL.INI or the INI file of your choice when you select the Apply button or press OK.

Each Protel layer is listed once in the left hand listbox, and each Layout layer is listed in the right hand listbox. For each Protel layer, you can select one or more Layout layers to map the information to upon translation. For instance, you would typically map the information on the TOP to Layout's TOP layer, and BOTTOM to Layout's BOTTOM layer.

You can also map more than one Protel layer to the same Layout layer.

Protel Layers

This is the list of layers which appears in the left hand column of the [PROTELXMAPPING] section of PROTEL.INI.

Add Protel Layer

You can type in a Protel layer name here to add it to the listbox. Then press <Enter> or the "Add" button to add the name to the list.

Delete Protel Layer

You can select a layer and press the "Delete" button to remove it from the list. This will remove the name entirely from the PROTEL.INI file when you select Apply or OK. If you wish to keep the layer name for reference but not have it mapped to anything, you have the option of removing all mappings from the layer instead.

Layout Layers

This is a list of the standard Layout layers. It uses the "Nickname" of the layer, which is a three letter abbreviation for

each of the layers. For instance, BOT is the bottom (solder) layer, SMT is the soldermask top layer, DRD is the drill drawing layer. The Layer spreadsheet in Layout lists the names and nicknames of all of the layers.

Note that you can select as many Layout layers as you wish to map from each Protel layer. If you deselect all Layout layers and press the "Edit Map" button, you remove the link between the selected Protel layer and Layout without actually removing the affected layers from the list.

Add Layout Layer

You can add your own Layout layer to the list using this edit box. Then press <Enter> or the "Add" button to add the name to the list.

Delete Layout Layer

This allows you to delete a Layout layer from the list. Note that all mappings that use that layer will be affected. If you delete the layer and don’t get a warning, it means that there were no more remaining mappings to that layer and it’s safe to delete it.

Edit Map

The Edit Map button creates the current layer mapping scheme in memory. However, the mapping scheme doesn’t get written to PROTEL.INI until you press OK or Apply. If you press Cancel, the edit will be aborted, and will not be written to PROTEL.INI.

Related topics

Protel PCB command (Import)

Protel PCB command (Export)

Protel99 SE command (Import)

Protel 99SE Import

The MAXP99X.EXE file translates a Protel 99 SE board file with a .PCB extension into an OrCAD Layout binary file with a .MAX extension.

You cannot import a board created in one application, such as Protel 99 SE, place and route it using Layout, then export it to an application other than Protel 99 SE—the import application must be the same as the export application.

To use the Protel translator

1. In Protel 99 SE, save your board file as an ASCII file with a .PCB extension.
2. From the Import menu in the session frame, choose Protel99 SE. The Protel 99SE to Layout MAX Translator dialog box appears.
3. Select the file (with a .PCB extension), then choose the OK button. The MAX Board dialog box appears.
4. Enter a filename with a .MAX extension, then choose the OK button. The Technology dialog box appears.
5. Select the P99SE.TCH file, then choose the OK button. The Protel Extractor dialog box appears.
6. When the message "Processing completed" appears, choose the OK button to dismiss the dialog box.

Protel 99SE Input/Output Files (Import)

Input files

There are three types of input files used by this translator:

- **Input Protel 99SE PCB File -** The input file from Protel 99 SE is the standard ASCII database.
- **Output Layout MAX File** -

- **Protel INI File -** The supplied file P99SE.INI holds the layer mapping and information that the Protel 99SE PCB to MAX translator uses to equate Protel layers with Layout layers. You can also create your own custom INI file by copying P99SE.INI and using the copy here. When you edit this information using the dialog, it gets written to the selected INI file for persistence.
- **Layout Technology/Template File -** The supplied file P99SE.TCH is an empty Layout database that contains the default Layout layer stackup, post processing defaults, and 16 empty via padstacks. You may create your own custom template file if you prefer, for instance, one containing a C sized sheet format.

Output files

The output file is a binary file in Layout's MAX format.

If you select "Overwrite existing file", the translator will allow you to overwrite an existing MAX file by the same name.

You can use the Browse buttons to find the files you need.

Protel 99SE Layer Mapping (Import)

The layer mapping tab allows you to easily map your Protel layers to the corresponding Layout layers.

The default layer mapping for most common Protel layer setups is provided in P99SE.INI, which is read into the layer mapping dialog for easy editing. Changes are saved to P99SE.INI or the INI file of your choice when you select the Apply button or press OK.

Each Protel layer is listed once in the left hand listbox, and each Layout layer is listed in the right hand listbox. For each Protel layer, you can select one or more Layout layers to map the information to upon translation. For instance, you would typically map the information on the TOP to Layout's TOP layer, and BOTTOM to Layout's BOTTOM layer.

You can also map more than one Protel layer to the same Layout layer.

Protel Layers

This is the list of layers which appears in the left hand column of the [P99XMAPPING] section of P99SE.INI.

Add Protel Layer

You can type in a Protel layer name here to add it to the listbox. Then press <Enter> or the "Add" button to add the name to the list.

Delete Protel Layer

You can select a layer and press the "Delete" button to remove it from the list. This will remove the name entirely from the P99SE.INI file when you select Apply or OK. If you wish to keep the layer name for reference but not have it mapped to anything, you have the option of removing all mappings from the layer instead.

Layout Layers

This is a list of the standard Layout layers. It uses the "Nickname" of the layer, which is a three letter abbreviation for each of the layers. For instance, BOT is the bottom (solder) layer, SMT is the soldermask top layer, DRD is the drill drawing layer. The Layer spreadsheet in Layout lists the names and nicknames of all of the layers.

Note that you can select as many Layout layers as you wish to map from each Protel layer. If you deselect all Layout layers and press the "Edit Map" button, you remove the link between the selected Protel layer and Layout without actually removing the affected layers from the list.

Add Layout Layer

You can add your own Layout layer to the list using this edit box. Then press <Enter> or the "Add" button to add the name to the list.

Delete Layout Layer

This allows you to delete a Layout layer from the list. Note that all mappings that use that layer will be affected. If you delete the layer and don't get a warning, it means that there were no more remaining mappings to that layer and it's safe to delete it.

Edit Map

The Edit Map button creates the current layer mapping scheme in memory. However, the mapping scheme doesn't get written to P99SE.INI until you press OK or Apply. If you press Cancel, the edit will be aborted, and will not be written to P99SE.INI.

Related topics

Protel99 SE command (Import)

CadStar Import

The MAXSTRX.EXE file translates a CadStar or a MAXI/PC file with a .CDI extension into an OrCAD binary file with a .MAX extension.

You cannot import a board created in one application, such as CadStar, place and route it using Layout, then export it to an application other than CadStar---the import application must be the same as the export application.

To use the CadStar translator

1 In the CadStar or MAXI/PC software, save your file as an ASCII file with a .CDI extension.

2 From the Import menu in the session frame, choose CadStar PCB. The CadStar ASCII Input dialog box displays.

3 Select the file with the .CDI extension, then choose the OK button. The MaxRoute Binary Output dialog box displays.

4 Enter a filename with a .MAX extension, then choose the OK button. The Technology dialog box displays.

5 Select the DEFAULT.TCH file, then choose the OK button. The Cadstar Extractor dialog box displays.

6 When the message "Processing completed" displays, choose the OK button to dismiss the dialog box.

CadStar Input/Output Files (Import)

Input files

There are three types of input files used by this translator:

- **Input CadStar CDI File -** The input file from CadStar should be output using the ASCII out utility.
- **CadStar INI File -** The supplied file CADSTAR.INI holds the layer mapping and information that the CDI to MAX translator uses to equate CadStar layers with Layout layers. You can also create your own custom INI file by copying CADSTAR.INI and using the copy here. When you edit this information using the dialog, it gets written to the selected INI file for persistence.
- **Layout Technology/Template File -** The supplied file CADSTAR.TCH is an empty Layout database that contains the default Layout layer stackup, post processing defaults, and 16 empty via padstacks. You may create your own custom template file if you prefer, for instance, one containing a C sized sheet format.

Output files

The output file is a binary file in Layout's MAX format.

If you select "Overwrite existing file", the translator will allow you to overwrite an existing MAX file by the same name.

You can use the Browse buttons to find the files you need.

CadStar Layer Mapping (Import)

The layer mapping tab allows you to easily map your CadStar layers to the corresponding Layout layers.

The default layer mapping for most common CadStar layer setups is provided in CADSTAR.INI, which is read into the layer mapping dialog for easy editing. Changes are saved to CADSTAR.INI or the INI file of your choice when you select the Apply button or press OK.

Each CadStar layer is listed once in the left hand listbox, and each Layout layer is listed in the right hand listbox. For each CadStar layer, you can select one or more Layout layers to map the information to upon translation. For instance, you would typically map the information on the Layer 1 to Layout's TOP layer, and Layer 16 to Layout's BOTTOM layer.

You can also map more than one CadStar layer to the same Layout layer.

CadStar Layers

This is the list of layers which appears in the left hand column of the [CADSTARXMAPPING] section of CADSTAR.INI.

Add CadStar Layer

You can type in a CadStar layer name here to add it to the listbox. Then press <Enter> or the "Add" button to add the name to the list.

Delete CadStar Layer

You can select a layer and press the "Delete" button to remove it from the list. This will remove the name entirely from the

CADSTAR.INI file when you select Apply or OK. If you wish to keep the layer name for reference but not have it mapped to anything, you have the option of removing all mappings from the layer instead.

Layout Layers

This is a list of the standard Layout layers. It uses the "Nickname" of the layer, which is a three letter abbreviation for each of the layers. For instance, BOT is the bottom (solder) layer, SMT is the soldermask top layer, DRD is the drill drawing layer. The Layer spreadsheet in Layout lists the names and nicknames of all of the layers.

Note that you can select as many Layout layers as you wish to map from each CadStar layer. If you deselect all Layout layers and press the "Edit Map" button, you remove the link between the selected CadStar layer and Layout without actually removing the affected layers from the list.

Add Layout Layer

You can add your own Layout layer to the list using this edit box. Then press <Enter> or the "Add" button to add the name to the list.

Delete Layout Layer

This allows you to delete a Layout layer from the list. Note that all mappings that use that layer will be affected. If you delete the layer and don’t get a warning, it means that there were no more remaining mappings to that layer and it’s safe to delete it.

Edit Map

The Edit Map button creates the current layer mapping scheme in memory. However, the mapping scheme doesn’t get written to CADSTAR.INI until you press OK or Apply. If you press Cancel, the edit will be aborted, and will not be written to CADSTAR.INI.

Related topics

Translating other file formats into Layout files

Creating a new Layout project from a Capture design

CadStar PCB command (Import)

CadStar PCB command (Export)

Tango Import

The MAXTANX.EXE file translates a Tango Series II file or a Tango-PCB PLUS file with a .PCB extension into an OrCAD binary file with a .MAX extension.

You cannot import a board created in one application, such as Tango, place and route it using Layout, then export it to an application other than Tango---the import application must be the same as the export application.

To use the Tango translator

1. In the Tango Series II or Tango-PCB PLUS software, save your file as an ASCII file with a .PCB extension.
2. From the Import menu in the session frame, choose Tango PCB. The Tango Input PCB dialog box displays.
3. Select the file (with a .PCB extension), then choose the OK button. The MAX Board dialog box displays.
4. Enter a filename with a .MAX extension, then choose the OK button. The Technology dialog box displays.
5. Select the DEFAULT.TCH file, then choose the OK button. The Tango Extractor dialog box displays.
6. When the message "Processing completed" displays, choose the OK button to dismiss the dialog box.

Tango Input/Output Files (Import)

Input files

There are three types of input files used by this translator:

- **Input Tango PCB File -** The input file from Tango is the standard ASCII database.
- **Tango INI File -** The supplied file TANGO.INI holds the layer mapping information that the PCB to MAX translator uses to equate Tango layers with Layout layers. You can also create your own custom INI file by copying TANGO.INI and using the copy here. When you edit this information using the dialog, it gets written to the selected INI file for persistence.
- **Layout Technology/Template File -** The supplied file TANGO.TCH is an empty Layout database that contains the default Layout layer stackup, post processing defaults, and 16 empty via padstacks. You may create your own custom template file if you prefer, for instance, one containing a C sized sheet format.

Output files

The output file is a binary file in Layout's MAX format.

If you select "Overwrite existing file", the translator will allow you to overwrite an existing MAX file by the same name.

You can use the Browse buttons to find the files you need.

Tango Layer Mapping (Import)

The layer mapping tab allows you to easily map your Tango layers to the corresponding Layout layers.

The default layer mapping for most common Tango layer setups is provided in TANGO.INI, which is read into the layer mapping dialog for easy editing. Changes are saved to TANGO.INI or the INI file of your choice when you select the Apply button or press OK.

Each Tango layer is listed once in the left hand listbox, and each Layout layer is listed in the right hand listbox. For each Tango layer, you can select one or more Layout layers to map the information to upon translation. For instance, you would typically map the information on the TOP to Layout's TOP layer, and BOTTOM to Layout's BOTTOM layer.

You can also map more than one Tango layer to the same Layout layer.

Tango Layers

This is the list of layers which appears in the left hand column of the [TANGOXMAPPING] section of TANGO.INI.

Add Tango Layer

You can type in a Tango layer name here to add it to the listbox. Then press <Enter> or the "Add" button to add the name to the list.

Delete Tango Layer

You can select a layer and press the "Delete" button to remove it from the list. This will remove the name entirely from the TANGO.INI file when you select Apply or OK. If you wish to keep the layer name for reference but not have it mapped to anything, you have the option of removing all mappings from the layer instead.

Layout Layers

This is a list of the standard Layout layers. It uses the "Nickname" of the layer, which is a three letter abbreviation for each of the layers. For instance, BOT is the bottom (solder) layer, SMT is the soldermask top layer, DRD is the drill drawing layer. The Layer spreadsheet in Layout lists the names and nicknames of all of the layers.

Note that you can select as many Layout layers as you wish to map from each Tango layer. If you deselect all Layout layers and press the "Edit Map" button, you remove the link between the selected Tango layer and Layout without actually removing the affected layers from the list.

Add Layout Layer

You can add your own Layout layer to the list using this edit box. Then press <Enter> or the "Add" button to add the name to the list.

Delete Layout Layer

This allows you to delete a Layout layer from the list. Note that all mappings that use that layer will be affected. If you delete the layer and don’t get a warning, it means that there were no more remaining mappings to that layer and it’s safe to delete it.

Edit Map

The Edit Map button creates the current layer mapping scheme in memory. However, the mapping scheme doesn’t get written to TANGO.INI until you press OK or Apply. If you press Cancel, the edit will be aborted, and will not be written to TANGO.INI.

Related topics

Translating other file formats into Layout files

Creating a new Layout project from a Capture design

Tango PCB command (Import)

Tango PCB command (Export)

SPECCTRA Import

You can use the SPECCTRA to Layout command on the File, Import submenu of the Layout session frame to translate SPECCTA files into Layout's .MAX file format.

To import SPECCTRA files into Layout

1. Activate the Layout session frame and choose the Import, SPECCTRA to Layout command from the File menu.
2. In the Input SPECCTRA File text box, enter the path and filename of the SPECCTRA file you want to translate.
3. In the Output Layout File text box, enter the path and filename of the SPECCTRA file you want to translate.
4. Select the Overwrite existing files option if you performed this operation previously and want to update an existing .MAX file.
5. Return to the Layout session frame and choose Open from the File menu.
6. Double-click on the .MAX file you just created.
7. Text box: Original Layout File

Related topics

SPECCTRA to Layout command (Import)

SPECCTRA to Layout dialog box

Layout to SPECCTRA dialog box

Layout to SPECCTRA command (Export)

GenCAD Import

To import GenCAD files into Layout

1. Activate the Layout session frame and choose GenCAD to Layout from the File, Import menu.
2. In the Input GenCAD File text box, specify a path and filename for the GenCAD file you want to translate.
3. In the Output Layout File text box, specify a path and filename for the Layout board file (.MAX) that you want to create by translating the specified GenCAD file.
4. Select the Overwrite existing files option if you performed this operation previously and want to update an existing .MAX file.
5. In the Input Technology text box, supply a name for the technology or layout file you want to use for the translation process. This file serves as a template for the .MAX file you create.
6. Option: Circumvent VeriBest bug.

GenCAD files produced by Veribest sometimes output solder side components with the padstacks and footprints flipped. If you are using the GenCAD format to input boards from Veribest, inspect the pads on the solder side to ensure they are not flipped.

Related topics

GenCAD to Layout dialog box

Layout to GenCAD dialog box

Layout to GenCAM dialog box

GenCAD to Layout command (Import)

Layout to GenCAD command (Export)

Layout to GenCAM command (Export)

PCB 386 Import

You can easily move a PCB 386+ design into Layout.

If translation problems arise, Layout displays error messages and writes them to a file in the output directory using the name of your output file and .ERR as an extension.

To import a PCB 386 design into Layout

1. From the File menu of the session frame, point to Import, then choose PCB 386+.
2. In the PCB 386 to MAX Translator dialog box, select the PCB 386+ board file, specify a directory for the output file, and enter a filename for the output file.
3. In the Output Layout File text box, enter the intended path and file name for the translation or use the Browse button to indicate where you want the file to reside. (The extension for a Layout library file is .LLB).

   If you do not want Layout to ask your permission before overwriting a file, select the Overwrite existing files option.
4. Choose the Translate button.

Notes about translating a PCB 386 board

Multiple-element pads

Layout does not support multiple-element (complex) padstacks. Only one of the defined elements is used as a pad; the others are converted to copper.

Open connections

After you translate a board from PCB 386+ to Layout, there may be open connections shown for what should be routed nets. This can be caused by duplicate wires situated on top of one another in the PCB 386+ file. This could happen on boards created with old versions if the routing was not changed in later versions.

The solution is to load the board file into PCB 386+, erase all routes and undelete. This eliminates duplicate wires. You need to then translate the board again.

Connection point outside pad dimensions

Layout does not support pads whose connection point is outside the pad dimensions. This can happen when large pad offsets are used or when a pad rotation rotates the pad off the connection point, or both.

Invalid arcs

Invalid arcs are converted to straight-line segments. This happens when one of the end points of the arc is the same as the center of the arc.

Wrong size thermals

Translated boards sometimes have thermals that are the wrong size. (A large thermal may get assigned when you wanted a small thermal.)

Translating a PCB 386 library

You can use your custom PCB 386+ libraries with Layout. In order to do so, you must first translate them.

In a translated library file, the module insertion point is set for use by pick-and-place machines and the module angle is set to zero. The layer colors of the translated file will match those of the PCB 386+ library if the files 386LIB.SF and 386LIB.TCH and are used to create the new library file.

To translate a PCB 386 library file

1. From the File menu of the Layout session frame window, choose the Import command, then choose PCB 386+.
2. In the PCB 386 to Max Translator dialog box, enter the name and path of the existing PCB 386+ library, or use the Browse button to locate the file. If you use the Browse button, you can choose .MLB in the Use Files of Type drop-down list.
3. Enter the path and file name for the translated file or use the Browse button to point to the location. The extension of a Layout library file is .LLB.
4. If you want Layout to prompt you before overwriting a file that has the name of the output file, be sure that the Overwrite existing files option is not selected
5. Choose the Translate button.

6 When Layout prompts you to use the 386LIB.SF and 386LIB.TCH files, choose the Yes button unless you have already selected a special strategy file and technology file you know contains the color setup you wish to use with your library. The 386LIB strategy and technology files are the same as the Layout 'std.sf' and 'default.tch' except that the color records have been altered to make the component outlines more visible and the drill drawing legend invisible in the translated library.

Related topics

PCB 386+ command (File menu, Import command)

PCB 386+ command (Export)

Translating other file formats into Layout files

Creating a new Layout project from a Capture design

Translating Layout files into other file formats

Related topics

System files

Design files

MAX ASCII files

Translating other file formats into Layout files

DXF import and export

Updating boards and libraries to Release 9 format

Converting pre-Layout v7.10 split planes

IDF Export

The IDF to Layout translator can be used to translate Layout board files (.MAX) into Intermediate Drawing Format (IDF). Note that IDF files come in pairs: the board file (.EMN) contains the board outline, keepouts, and component placements, while the library file (.EMP) contains additional data for each component placed in the board file, such as the component placement outline and the component height.

To translate Layout board files (.MAX) into IDF format

1. From the Layout session frame, choose the Layout to IDF command on the Export submenu (available via the File menu).
2. Choose the Browse button next to the Input Layout File text box.
3. Double-click on a .MAX file that you want to export. Layout automatically creates paths and filenames for the .EMN and .EMP files that are the products of translation.
4. Select the Overwrite existing files option if you wish to dispense with the warning Layout normally displays prior to overwriting files.
5. Option: Add a carriage return to each record

   Some 3-D board display tools require a carriage return after each record.
6. Option: Omit mechanical components

   When using the IDF as input to a thermal analysis tool, it is common to omit the mechanical components.
7. Option: Omit through-hole pad drills
8. Option: Use display units
9. Option: Use library footprints
10. Text box: Board Thickness
11. Text box: Default component height

Components that have no height definition will be assigned the default height.

Related topics

Layout to IDF dialog box

Layout to IDF command (Export)

IDF to Layout dialog box

IDF to Layout command (Import)

SPECCTRA Export

You can use the Layout to SPECCTRA command on the Export submenu of the File command (in the Layout session frame) to translate Layout board files (.MAX) into SPECCTRA file format.

To export Layout files in SPECCTRA format

1. Activate the Layout session frame and choose Layout to SPECCTRA from the Export submenu (available through the File menu).
2. Choose the Browse button beside the Input Layout File text box. Double-click on the icon of the Layout board file (.MAX) you want to translate. Layout automatically fills in the Input Layout File text box with the path and filename of the .MAX file you selected.
3. In the Output SPECCTRA File text box, Layout also automatically inserts a path and filename for the SPECCTRA file (.CCT) you want to create. You may change this value if you wish.

   Note: The .DSN SPECCTRA file extension is the same as a Capture design file extension.
4. Select the Overwrite existing files option if you performed this operation previously and want to update an existing SPECCTRA (.CCT) file.

5 Option: Include Advanced Per Layer/Object Spacing Rules
If you have the ADV option in your SPECCTRA autorouter.

6 Option: Create Do File template
If you have an existing .DO file, it will be compared to the skeleton file written by the translator. If you have edited the .DO file, the translator will warn you before overwriting it.

Related topics

SPECCTRA to Layout command (Import)

SPECCTRA to Layout dialog box

Layout to SPECCTRA dialog box

Layout to SPECCTRA command (Export)

GenCAD Export

To translate a Layout board file (.MAX) to GenCAD format

1 In the Layout session frame, choose File, then choose Import, and finally choose the Layout to GenCAD command.

2 Choose the Browse button beside the Input Layout File text box. The Input Layout MAX File dialog box appears.

3 Double-click on the icon of the Layout board file you want to translate. Layout automatically creates an entry for the Output GenCAD File text box.

4 Select the Overwrite existing files option to dispense with the warning Layout normally provides prior to overwriting existing files.

5 Option: Use library footprints

Related topics

GenCAD to Layout dialog box

Layout to GenCAD dialog box

Layout to GenCAM dialog box

GenCAD to Layout command (Import)

Layout to GenCAD command (Export)

Layout to GenCAM command (Export)

GenCAM Export

To export Layout .MAX files to GenCAM format

1. Activate the Layout session frame and from the Export submenu of the File menu, choose Layout to GenCAM.
2. Choose the Browse button beside the Input Layout File text box. Double-click on the Layout board file (.MAX) you wish to translate. Layout automatically creates a filename for the Output GenCAM File.
3. Text Box: Output GenCAM File
4. Overwrite existing files

Related topics

GenCAD to Layout dialog box

Layout to GenCAD dialog box

Layout to GenCAM dialog box

GenCAD to Layout command (Import)

Layout to GenCAD command (Export)

Layout to GenCAM command (Export)

IPC-356 Export

To export Layout .MAX files to IPC-356 format

1. In the Layout session frame, choose the Layout to IPC-356 command from the Export submenu of the File menu.
2. Choose the Browse button next to the Input Layout File text box. Double-click on the icon of the .MAX file you wish to translate. Layout automatically creates the path and name of the IPC-356 netlist file and inserts this information in the Output IPC356 Netlist File text box.
3. Select the Overwrite existing files if you wish to dispense with the standard warnings Layout displays prior to overwriting a file.
4. Option: Drills inside pads are through-holes
5. Include copper area with net names
   Layout outputs the net attributed copper areas as part of the netlist. Ensure you use this option if you have created customized shapes using copper area.
6. Settings group: Output Format
7. Option: Variable length records
8. Option: Fixed length records
   Ensure you use this option if the netlist is to be used by GerbTool.
9. Option: Fixed length records with line feed

Related topics

Layout to IPC-356 command (Export)

Layout to IPC-356 dialog box

Validating Gerber connectivity using an IPC-D-356 netlist

CadStar Export

CadStar Input/Output Files (Export)

Input files

There are three types of input files used by this translator:

- **Input Layout MAX File -** The input file from Layout, which must have been derived from an original CDI file from CadStar.
- **Original CadStar CDI file -** This is the original file that was used to create the MAX file before the placement and routing was done in Layout. If you do not have the original file, an equivalent one will still work, but you will get a warning in case you were unaware of the problem.
- **CadStar INI File -** The supplied file CADSTAR.INI holds the layer mapping and options/defaults information that the MAX to CDI translator uses to equate CadStar layers with Layout layers for transferring the route information back. You can also create your own custom INI file by copying CADSTAR.INI and using the copy here. When you edit this information using the dialog, it gets written to the selected INI file for persistence.

Output files

The output file is an ASCII file in Cadstar's CDI format.

If you select "Overwrite existing file", the translator will allow you to overwrite an existing CDI file by the same name. By default, the translator wizard adds "rou" to the filename in order to avoid overwriting the original file.

You can use the Browse buttons to find the files you need.

CadStar Layer Mapping (Export)

The layer mapping tab allows you to easily map your CadStar layers to the corresponding Layout layers.

The default layer mapping for most common CadStar layer setups is provided in CADSTAR.INI, which is read into the layer mapping dialog for easy editing. Changes are saved to CADSTAR.INI or the INI file of your choice when you select the Apply button or press OK.

Each CadStar layer is listed once in the left hand listbox, and each Layout layer is listed in the right hand listbox. For each CadStar layer, you can select one or more Layout layers to map the information to upon translation. For instance, you would typically map the information on the Layer 1 to Layout's TOP layer, and Layer 16 to Layout's BOTTOM layer.

CadStar Layers

This is the list of layers which appears in the left hand column of the [CADSTARBMAPPING] section of CADSTAR.INI. You should include any layers that contain routing information that you want sent back to CadStar from Layout.

Add CadStar Layer

You can type in a CadStar layer name here to add it to the listbox. Then press Enter or the "Add" button to add the name to the list.

Delete CadStar Layer

You can select a layer and press the "Delete" button to remove it from the list. This will remove the name entirely from the CADSTAR.INI file when you select Apply or OK. If you wish to keep the layer name for reference but not have it mapped to anything, you have the option of removing all mappings from the layer instead.

Layout Layers

This is a list of the standard Layout layers. It uses the "Nickname" of the layer, which is a three letter abbreviation for each of the layers. For instance, BOT is the bottom (solder) layer, SMT is the soldermask top layer, DRD is the drill drawing layer. The Layer spreadsheet in Layout lists the names and nicknames of all of the layers.

Note that you can select as many Layout layers as you wish to map from each CadStar layer. If you deselect all Layout layers and press the "Edit Map" button, you remove the link between the selected CadStar layer and Layout without actually removing the affected layers from the list.

Add Layout Layer

You can add your own Layout layer to the list using this edit box. Then press <Enter> or the "Add" button to add the name to the list.

Delete Layout Layer

This allows you to delete a Layout layer from the list. Note that all mappings that use that layer will be affected. If you delete the layer and don’t get a warning, it means that there were no more remaining mappings to that layer and it’s safe to delete it.

Edit Map

The Edit Map button creates the current layer mapping scheme in memory. However, the mapping scheme doesn’t get written to CADSTAR.INI until you press OK or Apply. If you press Cancel, the edit will be aborted, and will not be written to CADSTAR.INI.

Related topics

CadStar PCB command (Import)

CadStar PCB command (Export)

ODB++ Export

OrCAD Layout allows you to translate Layout board files (.MAX) and OrCAD ASCII format files (.MIN) to the Valor ODB++ or XML++ format.

Note: Before running Layout to ODB++ export, you must install the ODB++ Gateway to ODB++ Translator. You can download the ODB++ Gateway to ODB++ Translator from:

http://www.orcad.com/community.layout.dl.aspx

To export Layout files to Valor ODB++ or XML++ format

1. From the File menu of the session frame, point to Export, then choose Layout to ODB++.

   The Layout to ODB++ dialog box appears.

2. Enter the path and filename of the Layout board file (.MAX) or the OrCAD ASCII format file (.MIN) you want to translate to the Valor ODB++ or XML++ format, or click Browse to select the .MAX or .MIN file.

3. Enter the path to the directory where you want the Valor ODB++ or XML++ files to be created, or click Browse to select the directory.

4. Enter the name of the ODB++ or XML++ job to be created in the output directory.

   Layout creates a folder that has the same name as the job name in the output directory. The ODB++ or XML++ output files are located in this folder.

   Note: Valor requires that the job name is in lowercase and must not contain spaces. If you enter a job name in uppercase or with spaces, Layout changes it to lowercase and converts the spaces to underscore characters (_).

5 Select the Overwrite existing files check box if you do not want Layout to prompt you before overwriting existing files.

6 Select the Output copper pours check box if you intend to use the Valor Trilogy or Enterprise tool to create Gerber files from your ODB++ output files. You should always perform a connectivity check on your output files to verify correct translation of your design.

7 Select the output format as ODB++ or XML++.

8 Click the Translate button.

Layout creates a folder that has the same name as the job name in the output directory. The ODB++ or XML++ output files are located in this folder.

Related topics

Layout to ODB++ command (Export)

Layout to ODB++ dialog box

PADS Export

PADS Input/Output Files (Export)

Input files

There are three types of input files used by this translator:

- **Input Layout MAX File -** The input file from Layout, which must have been derived from an original asc file from PADS.
- **Original PADS ASCII file -** This is the original file that was used to create the MAX file before the placement and routing was done in Layout. If you do not have the original file, an equivalent one will still work, but you will get a warning in case you were unaware of the problem.

- **PADS INI File -** The supplied file PADS.INI holds the layer mapping and options/defaults information that the MAX to asc translator uses to equate PADS layers with Layout layers for transferring the route information back. You can also create your own custom INI file by copying PADS.INI and using the copy here. When you edit this information using the dialog, it gets written to the selected INI file for persistence.

Output files

The output file is an ASCII file in PADS's asc format.

If you select "Overwrite existing file", the translator will allow you to overwrite an existing asc file by the same name. By default, the translator wizard adds "rou" to the filename in order to avoid overwriting the original file.

You can use the Browse buttons to find the files you need.

PADS Layer Mapping (Export)

The layer mapping tab allows you to easily map your PADS layers to the corresponding Layout layers.

The default layer mapping for most common PADS layer setups is provided in PADS.INI, which is read into the layer mapping dialog for easy editing. Changes are saved to PADS.INI or the INI file of your choice when you select the Apply button or press OK.

Each PADS layer is listed once in the left hand listbox, and each Layout layer is listed in the right hand listbox. For each PADS layer, you can select one or more Layout layers to map the information to upon translation. For instance, you would typically map the information on the Layer 1 to Layout's TOP layer, and Layer 16 to Layout's BOTTOM layer.

PADS Layers

This is the list of layers which appears in the left hand column of the [PADSBMAPPING] section of PADS.INI. You should

include any layers that contain routing information that you want sent back to PADS from Layout.

Add PADS Layer

You can type in a PADS layer name here to add it to the listbox. Then press <Enter> or the "Add" button to add the name to the list.

Delete PADS Layer

You can select a layer and press the "Delete" button to remove it from the list. This will remove the name entirely from the PADS.INI file when you select Apply or OK. If you wish to keep the layer name for reference but not have it mapped to anything, you have the option of removing all mappings from the layer instead.

Layout Layers

This is a list of the standard Layout layers. It uses the "Nickname" of the layer, which is a three letter abbreviation for each of the layers. For instance, BOT is the bottom (solder) layer, SMT is the soldermask top layer, DRD is the drill drawing layer. The Layer spreadsheet in Layout lists the names and nicknames of all of the layers.

Note that you can select as many Layout layers as you wish to map from each PADS layer. If you deselect all Layout layers and press the "Edit Map" button, you remove the link between the selected PADS layer and Layout without actually removing the affected layers from the list.

Add Layout Layer

You can add your own Layout layer to the list using this edit box. Then press <Enter> or the "Add" button to add the name to the list.

Delete Layout Layer

This allows you to delete a Layout layer from the list. Note that all mappings that use that layer will be affected. If you delete the layer and don't get a warning, it means that there were no more remaining mappings to that layer and it's safe to delete it.

Edit Map

The Edit Map button creates the current layer mapping scheme in memory. However, the mapping scheme doesn't get written to PADS.INI until you press OK or Apply. If you press Cancel, the edit will be aborted, and will not be written to PADS.INI.

Related topics

PADS PCB command (Import)

PADS PCB command (Export)

P-CAD Export

P-CAD Input/Output Files (Export)

Input files

There are three types of input files used by this translator:

- **Input Layout MAX File -** The input file from Layout, which must have been derived from an original PDF file from P-CAD.
- **Original P-CAD PDIF file -** This is the original file that was used to create the MAX file before the placement and routing was done in Layout. If you do not have the original file, an equivalent one will still work, but you will get a warning in case you were unaware of the problem.

- **P-CAD INI File -** The supplied file P-CAD.INI holds the layer mapping and options/defaults information that the MAX to PDF translator uses to equate P-CAD layers with Layout layers for transferring the route information back. You can also create your own custom INI file by copying P-CAD.INI and using the copy here. When you edit this information using the dialog, it gets written to the selected INI file for persistence.

Output files

The output file is an ASCII file in P-CAD's PDF format.

If you select "Overwrite existing file", the translator will allow you to overwrite an existing PDF file by the same name. By default, the translator wizard adds "rou" to the filename in order to avoid overwriting the original file.

You can use the Browse buttons to find the files you need.

P-CAD Layer Mapping (Export)

The layer mapping tab allows you to easily map your P-CAD layers to the corresponding Layout layers.

The default layer mapping for most common P-CAD layer setups is provided in P-CAD.INI, which is read into the layer mapping dialog for easy editing. Changes are saved to P-CAD.INI or the INI file of your choice when you select the Apply button or press OK.

Each P-CAD layer is listed once in the left hand listbox, and each Layout layer is listed in the right hand listbox. For each P-CAD layer, you can select one or more Layout layers to map the information to upon translation. For instance, you would typically map the information on the Layer 1 to Layout's TOP layer, and Layer 16 to Layout's BOTTOM layer.

P-CAD Layers

This is the list of layers which appears in the left hand column of the [PCADBMAPPING] section of P-CAD.INI. You should

include any layers that contain routing information that you want sent back to P-CAD from Layout.

Add P-CAD Layer

You can type in a P-CAD layer name here to add it to the listbox. Then press <Enter> or the "Add" button to add the name to the list.

Delete P-CAD Layer

You can select a layer and press the "Delete" button to remove it from the list. This will remove the name entirely from the P-CAD.INI file when you select Apply or OK. If you wish to keep the layer name for reference but not have it mapped to anything, you have the option of removing all mappings from the layer instead.

Layout Layers

This is a list of the standard Layout layers. It uses the "Nickname" of the layer, which is a three letter abbreviation for each of the layers. For instance, BOT is the bottom (solder) layer, SMT is the soldermask top layer, DRD is the drill drawing layer. The Layer spreadsheet in Layout lists the names and nicknames of all of the layers.

Note that you can select as many Layout layers as you wish to map from each P-CAD layer. If you deselect all Layout layers and press the "Edit Map" button, you remove the link between the selected P-CAD layer and Layout without actually removing the affected layers from the list.

Add Layout Layer

You can add your own Layout layer to the list using this edit box. Then press <Enter> or the "Add" button to add the name to the list.

Delete Layout Layer

This allows you to delete a Layout layer from the list. Note that all mappings that use that layer will be affected. If you delete the layer and don't get a warning, it means that there were no more remaining mappings to that layer and it's safe to delete it.

Edit Map

The Edit Map button creates the current layer mapping scheme in memory. However, the mapping scheme doesn't get written to P-CAD.INI until you press OK or Apply. If you press Cancel, the edit will be aborted, and will not be written to P-CAD.INI.

Related topics

P-CAD PCB command (Import)

P-CAD PCB command (Export)

Protel Export

Protel Input/Output Files (Export)

Input files

There are three types of input files used by this translator:

- **Input Layout MAX File -** The input file from Layout, which must have been derived from an original PCB file from Protel.
- **Original Protel PCB file -** This is the original file that was used to create the MAX file before the placement and routing was done in Layout. If you do not have the original file, an equivalent one will still work, but you will get a warning in case you were unaware of the problem.

- **Protel INI File -** The supplied file PROTEL.INI holds the layer mapping information that the MAX to PCB translator uses to equate Protel layers with Layout layers for transferring the route information back. You can also create your own custom INI file by copying PROTEL.INI and using the copy here. When you edit this information using the dialog, it gets written to the selected INI file for persistence.

Output files

The output file is an ASCII file in Protel's PCB format.

If you select "Overwrite existing file", the translator will allow you to overwrite an existing PCB file by the same name. By default, the translator wizard adds "rou" to the filename in order to avoid overwriting the original file.

You can use the Browse buttons to find the files you need.

Protel Layer Mapping (Export)

The layer mapping tab allows you to easily map your Protel layers to the corresponding Layout layers.

The default layer mapping for most common Protel layer setups is provided in PROTEL.INI, which is read into the layer mapping dialog for easy editing. Changes are saved to PROTEL.INI or the INI file of your choice when you select the Apply button or press OK.

Each Protel layer is listed once in the left hand listbox, and each Layout layer is listed in the right hand listbox. For each Protel layer, you can select one or more Layout layers to map the information to upon translation. For instance, you would typically map the information on the TOP to Layout's TOP layer, and the MID_1 layer to Layout's INNER1 layer.

Protel Layers

This is the list of layers which appears in the left hand column of the [PROTELBMAPPING] section of PROTEL.INI. You

should include any layers that contain routing information that you want sent back to Protel from Layout.

Add Protel Layer

You can type in a Protel layer name here to add it to the listbox. Then press <Enter> or the "Add" button to add the name to the list.

Delete Protel Layer

You can select a layer and press the "Delete" button to remove it from the list. This will remove the name entirely from the PROTEL.INI file when you select Apply or OK. If you wish to keep the layer name for reference but not have it mapped to anything, you have the option of removing all mappings from the layer instead.

Layout Layers

This is a list of the standard Layout layers. It uses the "Nickname" of the layer, which is a three letter abbreviation for each of the layers. For instance, BOT is the bottom (solder) layer, SMT is the soldermask top layer, DRD is the drill drawing layer. The Layer spreadsheet in Layout lists the names and nicknames of all of the layers.

Note that you can select as many Layout layers as you wish to map from each Protel layer. If you deselect all Layout layers and press the "Edit Map" button, you remove the link between the selected Protel layer and Layout without actually removing the affected layers from the list.

Add Layout Layer

You can add your own Layout layer to the list using this edit box. Then press <Enter> or the "Add" button to add the name to the list.

Delete Layout Layer

This allows you to delete a Layout layer from the list. Note that all mappings that use that layer will be affected. If you delete the layer and don't get a warning, it means that there were no more remaining mappings to that layer and it's safe to delete it.

Edit Map

The Edit Map button creates the current layer mapping scheme in memory. However, the mapping scheme doesn't get written to PROTEL.INI until you press OK or Apply. If you press Cancel, the edit will be aborted, and will not be written to PROTEL.INI.

Related topics

Protel PCB command (Import)

Protel PCB command (Export)

Tango Export

Tango Input/Output Files (Export)

Input files

There are three types of input files used by this translator:

- **Input Layout MAX File -** The input file from Layout, which must have been derived from an original PCB file from Tango.
- **Original Tango PCB file -** This is the original file that was used to create the MAX file before the placement and routing was done in Layout. If you do not have the original file, an equivalent one will still work, but you will get a warning in case you were unaware of the problem.

- **Tango INI File -** The supplied file TANGO.INI holds the layer mapping information that the MAX to PCB translator uses to equate Tango layers with Layout layers for transferring the route information back. You can also create your own custom INI file by copying TANGO.INI and using the copy here. When you edit this information using the dialog, it gets written to the selected INI file for persistence.

Output files

The output file is an ASCII file in Tango's PCB format.

If you select "Overwrite existing file", the translator will allow you to overwrite an existing PCB file by the same name. By default, the translator wizard adds "rou" to the filename in order to avoid overwriting the original file.

You can use the Browse buttons to find the files you need.

Tango Layer Mapping (Export)

The layer mapping tab allows you to easily map your Tango layers to the corresponding Layout layers.

The default layer mapping for most common Tango layer setups is provided in TANGO.INI, which is read into the layer mapping dialog for easy editing. Changes are saved to TANGO.INI or the INI file of your choice when you select the Apply button or press OK.

Each Tango layer is listed once in the left hand listbox, and each Layout layer is listed in the right hand listbox. For each Tango layer, you can select one or more Layout layers to map the information to upon translation. For instance, you would typically map the information on the TOP to Layout's TOP layer, and on BOTTOM to Layout's BOTTOM layer.

Tango Layers

This is the list of layers which appears in the left hand column of the [TANGOBMAPPING] section of TANGO.INI. You should

include any layers that contain routing information that you want sent back to Tango from Layout.

Add Tango Layer

You can type in a Tango layer name here to add it to the listbox. Then press <Enter> or the "Add" button to add the name to the list.

Delete Tango Layer

You can select a layer and press the "Delete" button to remove it from the list. This will remove the name entirely from the TANGO.INI file when you select Apply or OK. If you wish to keep the layer name for reference but not have it mapped to anything, you have the option of removing all mappings from the layer instead.

Layout Layers

This is a list of the standard Layout layers. It uses the "Nickname" of the layer, which is a three letter abbreviation for each of the layers. For instance, BOT is the bottom (solder) layer, SMT is the soldermask top layer, DRD is the drill drawing layer. The Layer spreadsheet in Layout lists the names and nicknames of all of the layers.

Note that you can select as many Layout layers as you wish to map from each Tango layer. If you deselect all Layout layers and press the "Edit Map" button, you remove the link between the selected Tango layer and Layout without actually removing the affected layers from the list.

Add Layout Layer

You can add your own Layout layer to the list using this edit box. Then press <Enter> or the "Add" button to add the name to the list.

Delete Layout Layer

This allows you to delete a Layout layer from the list. Note that all mappings that use that layer will be affected. If you delete the layer and don't get a warning, it means that there were no more remaining mappings to that layer and it's safe to delete it.

Edit Map

The Edit Map button creates the current layer mapping scheme in memory. However, the mapping scheme doesn't get written to TANGO.INI until you press OK or Apply. If you press Cancel, the edit will be aborted, and will not be written to TANGO.INI.

Related topics

Tango PCB command (Import)

Tango PCB command (Export)

HyperLynx Export

To export Layout .MAX files to HyperLynx

1. Activate the Layout session frame. Choose the Layout to Hyperlynx command from the Export submenu (accessible via the File menu).
2. Choose Open from the File menu in the Layout "MAX" to "HYP" Translator window.
3. Double-click on the .MAX file you want to translate. The HyperLynx translator creates an .HYP file in the directory of the .MAX file you selected for translation.

Note: Only the components with less than 1000 pins are translated by Hyperlynx.

Related topics

Layout MAX to HYP window

Layout to Hyperlynx command (Export)

PCB 386+ Export

You can easily move a Layout design into PCB 386+.

If translation problems arise, Layout displays error messages and writes them to a file in the output directory using the name of your output file and .ERR as an extension.

To export Layout .MAX files to PCB 386+

1. From the File menu of the session frame, point to Export, then choose PCB 386+.

   The Layout to PCB386 dialog box appears.

2. Specify the path to the Layout .MAX file you want to translate, or click Browse to select the file.
3. Specify the path to the output PCB386 board file, or click Browse to select the file.
4. Select the Overwrite existing files check box if you do not want Layout to prompt you before overwriting existing files.
5. Click the Translate button.

Related topics

PCB 386+ command (Export)

PCB 386+ command (File menu, Import command)

DXF import and export

DXF import and export are ideally suited to translating a Layout board file to AutoCAD, adding mechanical or documentation drawings and text, then returning to Layout.

The MAXDXF.EXE file translates a drawing exchange format (.DXF) file and produces a Layout file with a .MAX extension, or translates a .MAX file to the .DXF format.

Related topics

System files

Design files

MAX ASCII files

Translating other file formats into Layout files

Translating Layout files into other file formats

Updating boards and libraries to Release 9 format

Converting pre-Layout v7.10 split planes

Importing a DXF file

To map a DXF file in which all entities are on a single layer

1 Create a MAXDXF.INI file which uses the AutoCAD layer 0 twice: once to translate all the polygons in the DXF file into obstacles and once to translate the circles into holes. For example:

```
[OBSTACLE]
TOP = 0
[MECHANICAL]
UNPLATED_MOUNTING_HOLES = 0
```

The polygons will all be on the Top layer along with one extra obstacle for each hole.

2 Select each obstacle and choose Properties from the pop-up menu. In the Edit Obstacle dialog box, select the target layer for the obstacle and select either Board Outline or Keepout from the Obstacle type drop-down list. You will need to look at the software for specific command and option names.

To merge separate DXF files into a single Layout layer

1 Copy your MAXDXF.INI file into the project directory and give it a unique name, maybe TEMP.

2 Edit the temporary file (in this example, "TEMP") and delete everything except the lines for the definitions and sections you need:

```
[MECHANICAL]
BOARD_OUTLINE = file1.dxf
COMP_KEEPOUT_ALL = file2.dxf
PLATED_MOUNTING_HOLES = file3.dxf
```

3 When MAXDXF prompts you for the name of the input DXF file, give it the temporary file name (in this case, "TEMP") instead. MAXDXF will read the three files listed in the example and translate them into the appropriate Layout objects within your output MAX file.

To import metric DXF files and create metric Layout board files from them

1 In your MAXDXF.INI file set the UNITS_DIVISOR to 25.4:

UNITS_DIVISOR=25.4

2 When you are prompted for a Technology templates, select a technology template that is in millimeters.

To interpret AutoCAD Traces

1 Edit the [COPPER] section of your MAXDXF.INI file to include those layers that have AutoCAD Traces you want to define as polygons.

2 Edit the [OBSTACLE] section of the file to include the layers that have AutoCAD Traces you want to define as lines.

3 If a layer contains both polygon and line AutoCAD traces, you must reference that layer twice (in both the [COPPER] and [OBSTACLE] sections) and then delete the undesired objects from within Layout.

Related topics

Layer Mapping for DXF translation

Adding text and obstacles with DXF import

Creating single-layer DXF files through the Post Processor

Creating multi-layer DXF files by exporting

Translating other file formats into Layout files

Layer Mapping for DXF translation

A DXF file is composed of lines, arcs, circles, and polylines. There is nothing intrinsic about any of these objects which distinguishes a polyline used as part of a board outline from a Polyline used as part of a keepout. When performing a DXF batch translation, Layout uses the convention that all DXF objects on a particular named layer are translated into the same kind of Layout object.

In the [MECHANICAL] section of the MAXDXF.INI file, there is a line which reads:

```
[MECHANICAL]
BOARD_OUTLINE = MY_BOARD_OUTLINE_LAYER
```

This line tells the translator that all of the lines, arcs, and polylines in the DXF file, on the MY_BOARD_OUTLINE_LAYER are to be joined and translated into a board outline in your Layout .MAX file.

Objects for multiple layers

For Layout objects which can exist on multiple layers, there are special keywords in the MAXDXF.INI file to assist you. To translate the lines, arcs, and polylines in your DXF file on the MY_ROUTE_KEEPOUT_TOP layer into a Route-Keepout object on the TOP layer in Layout, you would put this line in your MAXDXF.INI file:

```
ROUTE_KEEPOUT_TOP = MY_ROUTE_KEEPOUT_TOP
```

The keyword directive on the left of the equal sign must be spelled exactly as shown. On the right side of the equal sign, you place the name of the appropriate layer within your DXF file.

These other directives perform the same function for other kinds of Layout objects:

```
ROUTE_KEEPOUT_BOT = MY_ROUTE_KEEPOUT_BOT
ROUTE_KEEPOUT_ALL = MY_ROUTE_KEEPOUT_ALL
NO_VIA_TOP = MY_NO_VIA_TOP
NO_VIA_BOT = MY_NO_VIA_BOT
NO_VIA_ALL = MY_NO_VIA_ALL
HEIGHT_RESTRICTED_TOP = MY_HEIGHT_RESTRICT_TOP
HEIGHT_RESTRICTED_BOT = MY_HEIGHT_RESTRICT_BOT
COMP_KEEPOUT_ALL = MY_KEEPOUT_ALL
COMP_KEEPOUT_TOP = MY_KEEPOUT_TOP
COMP_KEEPOUT_BOT = MY_KEEPOUT_BOT
COMP_GROUP_KEEPIN_ALL = MY_GROUP_KEEPIN_ALL
COMP_GROUP_KEEPIN_TOP = MY_GROUP_KEEPIN_TOP
COMP_GROUP_KEEPIN_BOT = MY_GROUP_KEEPIN_BOT
```

Plated and unplated mounting holes

Circles in your DXF can be translated into plated or unplated mounting holes using these two directives. In this example, circles on the DXF layer MY_PLATED_HOLES are translated into plated mounting holes, while circles on layer MY_MOUNTING_HOLES are translated as unplated holes.

```
PLATED_MOUNTING_HOLES = MY_PLATED_HOLES
UNPLATED_MOUNTING_HOLES = MY_MOUNTING_HOLES
```

The [OBSTACLE] section in the MAXDXF.INI file will allow you to translate all of the lines, arcs, circles, and polylines on any particular DXF layer into Layout obstacles on a specific target layer. Once they have been translated into Layout objects, you can edit the obstacles within Layout to complete the translation.

Some Mechanical CAD packages like AutoCAD support an object called a Trace. This is a solid polygon with either 3 or 4 vertices. If you have a company logo drawn from Traces, use the [COPPER] section of the MAXDXF.INI file to define the layer on which you would like the logo translated. Each Trace object in the DXF file will then be translated into a copper area within Layout.

When you translate a DXF interactively using Visual CADD (by pressing the Interact button in the DXF translation dialog), you have the opportunity to select one or more objects in your DXF, press the translate toolbar button in Visual CADD, interactively specify what kind of object you would like created from these DXF entities, and specify on which Layout layer you want to place it. On the other hand, if your Mechanical Design group has a particular layer naming convention that they follow, you can record that convention in your MAXDXF.INI file and simply perform a batch translation in one step.

The board outline is always placed on the Global Layer.

Related topics

Importing a DXF file

Adding text and obstacles with DXF import

Creating single-layer DXF files through the Post Processor

Creating multi-layer DXF files by exporting

Adding text and obstacles with DXF import

To add text and obstacles to a board design file

1. Create the board file in Layout.
2. From the session frame's Export menu, choose Layout to DXF.
3. Select the design file, then choose the Open button.
4. Enter the name of the .DXF file, then choose the Save button.
5. Edit the .DXF file in the application of your choice and save it.
6. Return to Layout's session frame, and from the Import menu, choose DXF to Layout. The AutoCAD DXF Input dialog box displays.
7. Choose the Browse button and select the .DXF file you want to translate.

   Layout automatically creates a filename for the output .MAX file. You may override the created filename if you want.
8. If you are creating a new design from your .DXF file, choose an appropriate technology file. If you are updating an existing design, enter the name of the .MAX file you are updating, as your technology file.
9. If you want to:

translate the .DXF file into Layout	click the Translate button
load the .DXF file into Visual CADD and interactively select and translate .DXF entities manually	click the Interact button

Note: Translation between .MAX and .DXF files requires that the file MAXDXF.INI be in the LAYOUT directory.

Note: If you are merging multiple DXF files with data in your .MAX file, do not select the Remove existing obstacles and text option in the DXF Import dialog box. If you want to replace the board outline and its annotations with entities from the .DXF file, select the Remove existing obstacles and text option.

Related topics

Translating other file formats into Layout files

Importing a DXF file

Layer Mapping for DXF translation

Creating single-layer DXF files through the Post Processor

Creating multi-layer DXF files by exporting

Creating a new Layout project from a Capture design

Creating single-layer DXF files through the Post Processor

This procedure creates a DXF file for each layer. After creating the DXF files, you can annotate the DXF files with IntelliCAD.

1. In Layout, from the Options menu, choose Post Process Settings. The Post Process spreadsheet appears.
2. Locate the rows for the layers that you wish to annotate. Typically AST, ASB and FAB layers are used to include specifications from IPC or ANSI documentation.
3. Double click in the Device column for the selected layer. The Post Process Settings dialog appears.
4. In the Format group box, check DXF.
5. In the Options group box, check Enable for Post Processing and click OK.
6. Repeat steps 3 and 4 for each layer you wish to annotate.
7. Close the Post Process spreadsheet.

8 From the Auto menu choose Run Post Processor. The post processor creates DXF files for the selected layers. These files are placed in the same directory as your MAX file, and are named with the extension of the selected layer. For example, .AST or .ASB.

9 For IntelliCAD to be able to open the DXF files you just created, the files must have a .DXF extension. In the Windows Explorer, add .DXF *after* the original extension. This allows you to differentiate between the various DXF layer files.

10 You can now open the new DXF files and add any desired annotations or dimension objects.

Related topics

Importing a DXF file

Layer Mapping for DXF translation

Adding text and obstacles with DXF import

Creating multi-layer DXF files by exporting

Creating multi-layer DXF files by exporting

This procedure creates one DXF file that contains multiple layers that can be individually annotated in IntelliCAD.

1 In the Layout session window, point to Export and choose Layout to DXF. The MAX to DXF dialog appears

2 In the DXF ini File text box, note the location of the current MAXDXF.INI file.

3 Launch Notepad or another text editor, and open MAXDXF.INI

4 Scroll down to the [LAYER] section.

5 Place a semicolon (;) at the beginning of all layer names that you do not want to include in the DXF file. Do not comment out any other information in this file.

6 Save the file in a new directory with a name that identifies the changes you have made. The file must continue to be named MAXDXF.INI. It is not recommended that you write over the original MAXDXF.INI file. For example:

..\Layout\maxdxf_assembly_only\maxdxf.ini

7 In the MAX to DXF dialog, locate the DXF ini File text box. Click the Browse button, locate the modified MAXDXF.INI file and click Open.

8 From the Input Layout File text box, click the Browse button, locate the MAX file to convert, and click Open. The file path is placed in the Input Layout File text box, and the Output to DXF text box is filled with the default DXF file name.

9 Check the Projection Plots check box.

or

If you intend to use the DXF file with a thermal modeling tool, clear this check box.

10 Click Translate to create the multi-layer DXF file. You can now open the new DXF file in IntelliCAD and add any desired annotations or dimension objects.

Related topics

Importing a DXF file

Layer Mapping for DXF translation

Adding text and obstacles with DXF import

Creating single-layer DXF files through the Post Processor

Updating boards and libraries to Release 9 format

Updating occurs automatically for any board or library loaded into Layout Release 9, or the library manager, and saved. If the board or library is saved under a different name, only the newly named file will be in Layout Release 9 format.

UPDATE90.EXE migration utility

We provide an UPDATE90.EXE utility that you can use to migrate entire directories from earlier versions of Layout to Layout Release 9 format. UPDATE90.EXE is on the product CD and, if you are going to use it, must be copied to your Layout directory. It uses the same DLLs and file access methods used by Layout, and provides the same results as loading an older file into Layout and saving it.

In addition, UPDATE90.EXE has options that you can use to:

- Reorder your component pads into ascending numeric order.
- Reset the component origin to pad 1.
- Reset the insertion point for surface-mount parts to the centroid of the part.
- Reset the insertion point for through-hole parts to either pad 1 or the centroid.
- Selectively control which components are updated.
- List filenames and footprint names rather than perform update actions, and import the list into a spreadsheet, using a comma-delimited format.
- Repair designs with the extraneous connection problem created by the releases of Layout Release 9 and Layout Release 9a.

Batch updating

If desired, you can update boards and libraries in a batch fashion using the UPDATE90.EXE migration utility. This is strongly recommended for custom libraries so they will load more efficiently and not get further out of date with later releases. As a safety precaution, it is recommended that you back up the boards and libraries prior to using UPDATE90.EXE.

To update all the libraries in your directory:

1. Double-click on UPDATE90.EXE in your Windows Explorer.
2. In the Update to Layout 9.00 dialog box, click the Browse button associated with the Input Layout File text box.
3. In the Input File dialog box, make your library directory the current directory, change the Files of type to Library Files (*.LLB), choose one of the libraries, then click Open.
4. After returning to the Update to Layout 9.00 dialog box, in the Input Layout File text box, change DRIVE:\PATH\FILENAME.LLB to DRIVE:\PATH*.LLB (changing the root filename to an asterisk).
5. Click the Update button.

When UPDATE90 sees meta-characters in the input file specification (an asterisk or a question mark), it ignores the output filename specification, except for the ending suffix. When you click the Update button, all files that match your input specification are updated and written back to the source directory, using the suffix off the output file specification (usually .NEW) in place of the original .LLB suffix.

Even though your updated libraries have a suffix other than .LLB, you can inspect the updated libraries using the Layout library manager. When you are satisfied with the success of the update operation, you can delete the old files (DEL *.LLB) and then rename the new files (REN *.NEW *.LLB) using a DOS window.

Related topics

System files

Design files

MAX ASCII files

Translating other file formats into Layout files

Translating Layout files into other file formats

DXF import and export

Converting pre-Layout v7.10 split planes

Converting pre-Layout v7.10 split planes

Designs created with versions prior to v7.10 must be updated properly so that the Release 9 split plane functionality works properly.

Note: If your pre-Layout v7.10 design is not update properly, your Layout Release 9 and later designs will not have correct split planes.

In v7.10 or later, planes are split by drawing one or more copper pours over the regions you want to dedicate to specific nets. You can assign only one net (the dominant net) to the plane itself. Other nets are assigned to the copper pours. After assigning nets, you can route and place vias for them over any region of the board.

For through-hole vias, Layout v7.10 or later takes care to create thermal reliefs only within the appropriate regions. If you have Use Pours for Connectivity selected in the User Preferences dialog box, Layout v7.10 and later updates ratsnests automatically.

Split planes in Layout versions prior to v7.10 were created differently, so boards with split planes being migrated to v7.10 and later require some small changes to the design.

When discussing the necessary changes, the following assumptions are made:

- Split planes in versions prior to v7.10 have two nets assigned to the same plane layer (the split plane).
- The plane is electronically broken into two pieces, either by free tracks, or board detail around the region dedicated to the second net.
- Routes for the two nets are constrained so that through-hole vias for each net occur over regions belonging only to that net.

To convert a pre-v7.10 split plane to a v7.10 and later split plane:

1. Unassign the second net from the plane layer.
2. Convert the free track or detail into a copper pour zone that encompasses the entire area of the second net.
3. Select Use Pours for Connectivity in the User Preferences dialog box to eliminate the ratsnest display of the two nets.

Related topics

System files

Design files

MAX ASCII files

Translating other file formats into Layout files

Translating Layout files into other file formats

DXF import and export

Updating boards and libraries to Release 9 format

Setting up the board

5

In Layout, you should set up the board before you begin placing components. This chapter explains how to set up a board by combining a board template or a technology template with other Layout commands and processes.

The steps involved in the board setup process are listed below, but not all of them are necessary for every board.

- Creating a new Layout project from a Capture design
- Load a template
- Create a board outline
- Set the units of measurement
- Set system grids
- Add mounting holes
- Define the layer stack
- Set global spacing
- Define padstacks
- Define vias
- Set net properties

Creating a new Layout project from a Capture design

Preparing a Capture design for Layout is a two-part process. First, you must create a valid design and then create a netlist in an .MNL format for Layout. After you have prepared your Capture design, you can create a new Layout design using the .MNL netlist.

You can bring Capture netlist information into Layout in two ways. You can choose one of the AutoECO options to merge the netlist with the board file, or you can select the Run ECO to Layout option in Capture (in the Create Netlist dialog box) to automatically communicate modifications to Layout. If the board file is open when you update the netlist file, Layout automatically displays a dialog box asking if you want to load the new netlist file. If the board file is not open when the netlist changes, Layout prompts you to load the modified netlist when you re-open the board file.

Capture's Layout macros

Capture comes with three configured macros. These macros were designed to assist you in preparing a design for Layout. The macros are as follows:

Set Layout Part Properties

Use this command to add Layout specific properties to the selected parts. Capture displays the Set Layout Part Properties dialog box for you to provide values for Layout properties. When you choose the OK button, Capture adds the Layout properties and the values you provided to the selected parts. The properties that were not supplied values for are not added to the selected parts.

Set Layout Pin Properties

Use this command to add Layout specific properties to the selected pins. Capture displays the Set Layout Pin Properties dialog box for you to provide values for Layout properties. When you choose the OK button, Capture adds the Layout

properties and the values you provided to the selected pins. The properties that were not supplied values for are not added to the selected pins.

Set Layout Net Properties

Use this command to add Layout specific properties to the selected nets. Capture displays the Set Layout Net Properties dialog box for you to provide values for Layout properties. When you choose the OK button, Capture adds the Layout properties and the values you provided to the selected nets. The properties that were not supplied values for are not added to the selected nets.

Related topics

Back annotating

AutoECO

Using technology templates

When creating a new board design, Layout asks you for a template file to use. There are two kinds of templates that you can indicate for use:

- technology (.TCH)
- board (.TPL)

Technology templates let you establish design standards and manufacturing complexity. Layout comes with several technology templates to choose from, including a default template. See Technology templates on page 127 for a detailed discussion.

Note: When you load a new technology template, some existing board data is overwritten, and some is ignored.

A board template contains physical board objects such as a board outline, mounting holes, and so on. Layout comes with several board templates to choose from.

Loading a technology template

You can load either type of template when creating your design. If you load a board template, the default technology template is included. If the default technology template does not satisfy your design standards and manufacturing complexity requirements, you can load one that does.

Note: For a complete list and detailed description of technology templates included with Layout, see Technology templates on page 127.

If you load only a technology template, there will be no physical board objects upon opening.

To load a template

1. From the File menu, choose Load. The Load File dialog box appears.
2. Select either a board (.TPL) or technology (.TCH) template.
3. Click the Open button.

Creating custom technology templates

It is easiest to create a custom template by modifying an existing board template and saving it under a new name, but you can also start with an empty board file. You can use your custom template with any Layout board.

You may want to create a custom template if, for instance, you want to use a board outline provided with Layout, but need more from the technology template than DEFAULT.TCH can offer. Or, you may want to create a custom template if you are creating your own board outline. If you know that you will use the board outline in future boards, you can create a custom template that incorporates the outline and any other design rules you use often.

To create a custom template using one of Layout's board outlines

1. From the File menu, choose Open. The Open Board dialog box appears.
2. Change Files of type to All Files, open the DATA folder and select the board template (.TPL) that has the board outline you want to use, then choose the Open button. The board template opens in Layout.
3. From the File menu, choose Load. The Load File dialog box appears.
4. Change Files of type to Template, select the technology template (.TCH) you want to use, then choose the Open button. Layout loads the technology file.
5. Define other board criteria as necessary using the processes in this chapter.
6. From the File menu, choose Save As. The Save File As dialog box appears.
7. Change Save as type to Template, select which folder to save the file in, supply a filename (with a .TPL extension), then choose the Save button.

To create a custom template using your own board outline

1. From the File menu, choose New. The Load Template File dialog box appears.
2. Choose the Cancel button. An empty board opens in the design window.
3. From the View menu, choose Zoom All. The entire board (its DRC box and drill chart) appears in the design window.
4. Create a board outline by following the instructions in *Creating a board outline* in this chapter.
5. From the File menu, choose Load. The Load File dialog box appears.

6 Change Files of type to Template, select the technology template (.TCH) you would like to save with the new board outline, then choose the Open button. Layout loads the technology file.

7 Define other board criteria as necessary using the processes in this chapter.

8 From the File menu, choose Save As. The Save File As dialog box appears.

9 Change Save as type to Template, select which folder to save the file in, supply a filename (with a .TPL extension), then choose the Save button.

To create a custom template from an existing board

1 Open the board you want to use as a basis for the template.

2 Choose the spreadsheet toolbar button, then choose Components. The Components spreadsheet appears.

3 Select all the components you want to remove (excluding those you want in the template, such as mounting holes, preplaced connectors, and so on), then press the DELETE key.

4 In the Edit Component dialog box, select the Not in Netlist option for those items (such as mounting holes) that will not be in your netlist. (Double-click on an item in the Components spreadsheet to display the Edit Component dialog box.)

5 Choose the spreadsheet toolbar button, then choose Nets. The Nets spreadsheet appears.

6 Select all the nets in the spreadsheet, then press the DELETE key.

7 From the File menu, choose Save As. The Save File As dialog box appears.

8 Change Save as type to Template, select which folder to save the file in, supply a filename (with a .TPL extension), then choose the Save button.

Creating a board outline

The board outline defines the boundary of the board. Board outlines are created using the Obstacle tool and the Edit Obstacle dialog box. After creating the board outline, you can save it to a technology template for use in future designs.

Note: Layout requires exactly one board outline, on the global layer.

To create a board outline

1. From the Tool menu, choose Dimension, then choose Datum. Click on the lower left corner of the board outline to place the datum (to provide a starting grid for component placement). Press HOME to redraw the screen.

   Note: Placing the datum in the lower-left corner of the board outline gives you positive X, Y coordinates, while placing it in other corners gives you negative coordinates (in your reports and post processing results).

   Note: Because the board datum is used for all grids, if you move the datum after component placement, your place, routing, and via grids will all be affected. And, you may have difficulty replacing the datum at the precise location you moved it from.

2. Choose the obstacle toolbar button.

3. From the pop-up menu, choose New, then from the pop-up menu, choose Properties. The Edit Obstacle dialog box appears.

4. From the Obstacle Type drop-down list, select Board outline.

5. In the Width text box, enter a value for the outline's width.

   Note: Layout has a 50 mils default board outline width, in order to provide clearance on plane layers for the copper of the plane to the edge of the board. One-half of the width is the pullback (25 mils in the default width), so set the board outline's width to two times the pullback you would like. The cut is made down the center of the board

outline obstacle.

6 From the Obstacle Layer drop-down list, select Global Layer, then choose the OK button. The Edit Obstacle dialog box closes.

7 Move to the point on the board at which you want to start drawing the outline, then click the left mouse button to insert the first corner.

Because a board outline must be a closed polygon, Layout automatically begins forming a closed area after you insert the first corner of the board outline, and automatically closes the polygon for you if you don't close it yourself.

8 Continue clicking the left mouse button to insert corners.

Note: If you zoom in while drawing, you can press C to put your current cursor location in the center of the screen. Hold down C and move the mouse to pan around the board.

9 After you click to insert the last corner, choose Finish from the pop-up menu. Layout automatically completes the board outline.

Moving the datum

The location of datum is constrained to the Place grid. You may need to change the grid in order to place the datum at a precise location. It is suggested that you put your datum in the lower left corner of the board design.

To move the datum

1 From the Tool menu, point to Dimension and choose Move Datum. Your pointer changes to a small cross.

2 Position the pointer at the point where you want the datum to appear. While you are moving the pointer, the coordinates at the bottom of the screen reflect the pointer position relative to the previous datum.

3 Click the left mouse button to place the datum. The coordinates in the status bar reflect distances relative to the new datum point. The datum tool is dismissed as soon as the new datum is established.

4 Click the right mouse button and choose End Command, or press the ESC key to exit the Move Datum mode.

Setting units of measurement

Layout has the ability to use either the metric or the inch system of measurement to specify distances and areas on your PCB. You can choose the units to use by selecting options from the System Settings dialog box.

Layout uses a very flexible and accurate scheme to insure 100% accuracy within either inch or metric designs that guarantees no significant errors as a result of any conversions between units.

The basis for this flexibility is Layout's ability to use whatever base unit is most reasonable for the given technology. The standard Layout base units are:

1/60th of a mil for inch
1/100th of a micron for metric

If you are using a standard inch design (derived from DEFAULT.TCH) and you choose Millimeters for your display unit, you will see the closest conversion for each of the inch base units expressed in millimeters. Conversely, if you are using a metric design (derived from METRIC.TCH), and you choose Inches for your display unit, you will see the closest conversion for each of the metric base units expressed in inches. Neither of these approaches actually converts any of the data. It simply allows you to view the data using different units.

The same applies to footprint libraries. A library derived from EMPTY.LLB will contain inch parts that can be viewed in either measuring system. A library derived from METRIC.LLB will contain metric parts that can be viewed in either measuring system.

If you select the Convert database option on the Display Units dialog box, on the other hand, Layout performs a complete conversion of the underlying base units from the inch version to the metric version or vice versa.

Note: There may be a one-time non-cumulative loss of precision of less than 1/100th of a mil when going from metric to inch. This is insignificant for manufacturing purposes and will not affect artwork generation.

In order to avoid cumulative conversion errors, you should always build parts in native metric units, and then, if necessary, convert those parts to inch units only *after* creation is completed. That way, any round-off errors will be non-cumulative, and limited to the non-significant range of less than 1/100th of a mil.

To build a part using true metric base units, you can either begin by loading the METRIC.LLB file from the LIBRARY directory, or begin with the file EMPTY.LLB, choose a metric unit (microns, millimeters, or centimeters) and choose the Convert database option on the Units dialog box. You now have a zero-error library file to build your part.

If you will then be working completely in metric, you need only make one change to the default setup of the LAYOUT.INI file.

Using a text editor like Notepad, or using the Text Editor command from the File menu in Layout, search for the string SCHGLOBAL in LAYOUT.INI (this file is in your Windows directory). The section referred to appears below:

```
[SCHGLOBAL]
USERDIV=60
#For standard OrCAD Technology Templates use:
#USERDIV=60
#Each user unit equals one mil.
#
#For OrCAD metric and small base unit
# (like METRIC.TCH, HYBRID.TCH or MCM.TCH)
#"TCH" files use:
#USERDIV=1000
#Each user unit equals one micron.
#
#Occasionally files will use other
# base units as well.
#If in doubt, use OrCAD Interchange (MIN)
# to create an ASCII file from your
# technology or MAX file, and check the
# field called "USERDIV" in the header.
```

Change the uncommented USERDIV=60 (shown in bold above) to USERDIV=1000. This tells Layout that the units being passed in from the schematic are going to be in microns rather than mils.

Remember, this only applies if your board design file is metric, but not if your library is metric and your board is in inches. If you use a metric library or a combination of metric and inch libraries with an inch printed circuit board design, leave this value at its default of 60. If you use metric or inch parts with a metric printed circuit board design, you will never have any errors in conversion.

If you use metric or inch parts with an inch printed circuit board design, all of the conversions necessary to use these parts without creating manufacturing errors are accounted for automatically when you load the part into your inch database.

Note: If your board uses metric units, you can achieve the best precision by using the METRIC.TCH technology template. With your board open in Layout, choose Load from the File menu, select METRIC.TCH, then choose the Open button. After METRIC.TCH loads, save your board.

To set measurement units

1. Open your board in Layout.
2. From the Options menu, choose System Settings. The System Settings dialog box appears.
3. Select mils, inches, microns, millimeters, or centimeters.
4. Choose the OK button.

Note: Once you decide on a measurement unit, you should stick with it and not change it in either your board or your schematic. If you back annotate to your schematic, then change to another measurement unit, it may cause board corruption problems.

Related topics

System Settings command

System Settings dialog box.

Setting system grids

Using the System Settings dialog box, you can set five distinct grid settings. The grid values that you assign determine the resolution of the pointer location coordinates given in the status bar in the lower left corner. For example, if the obstacle tool is selected and the Place grid is set to 100 mils, the coordinates that display are accurate to 100 mils.

Grid values are in user-specified units that you set in the Display Units group box in the System Settings dialog box. If you want to use fractions in your grid values, enter a space character following the integer and use a forward slash as the division character (for example, 8 1/3). You can also use decimals for rational numbers.

Note: Here are some rules of thumb for setting the grids:

For efficient routing performance, the routing grid and via grid should have the same value.

The place grid must be a multiple of the routing and via grids.

The routing grid should never be less than 5 mils.

The detail grid can be set as low as 1 mil for better resolution.

Components are placed on the place grid using the component datum, which is typically pad 1 (unless the component has been modified).

To set system grids

1 From the Options menu, choose System Settings. The System Settings dialog box appears.

2 Set these options, then choose the OK button.

Visible grid

Assigns a display grid based on the X and Y coordinates (for example, if you're using mils, a setting of 200 would place a grid dot at every 200 mils).

Detail grid

Assigns a drawing grid (for lines and text) based on the X and Y coordinates.

Place grid

Assigns a component placement grid based on the X and Y coordinates. For greatest routing efficiency, this value needs to be a multiple of the routing grid. The datum, or origin, of footprints is constrained to this grid.

Routing grid

Assigns a grid used for routing (see the routing grid chart below for suggested settings).

Via grid

Assigns a grid upon which you or the router can place vias.

The following chart is a synopsis of routing grids and how to use them in Layout.

Routing grid	Uses
Compatible grids 25, 12 1/2, 8 1/3, and 6 1/4:	
25, 12 1/2	Use for less dense (usually .45 density or greater) through-hole and SMT boards, and for routing one track between IC pins.
8 1/3	Use for a secondary grid on through-hole boards, and for a primary grid on SMT boards. Use as a secondary grid with 25 mils grid only if the 25 mils grid initially routes 95% or better.
6 1/4	Use for 6/6 technology, or denser one-between boards.
Compatible grids 20 and 10:	
20	Use for through-hole boards only. This is the most efficient way to route two tracks between IC pins.
10	Use for through-hole, two-between boards placed on a 50 mils grid, and for SMT boards using 10/10 technology. Also, use for special cases when a 20 mils grid causes off-grid jogs.
Compatible grids 25, 20, and 10:	
5	Use for extremely dense SMT boards that use 5 mils spacing and 5 mils track width (for mixed inch and metric technologies).

Note: Incompatible grids (such as 20 and 25) should not be mixed on the same board. If you find it necessary to do so, use a 5 mils grid for the final reroute pass.

Note: Also, a via grid smaller than the routing grid (for instance, a 5 mils via grid on a 25 mils grid board) increases completion on difficult SMT boards. Of

course, if a board is very dense, via sizes should by reduced to the minimum size possible, since vias are responsible for much of the channel blockage during routing.

Adding mounting holes to a board

You can add mounting holes to your board, and you can save them in a board template (.TPL). After adding the mounting holes to the board, define them as Not in Netlist. You can still attach these mounting holes to the ground net. The Not in Netlist flag keeps the ECO process from removing them.

To add mounting holes to your board

1. Choose the component toolbar button.
2. From the pop-up menu, choose New. The Add Component dialog box appears.
3. Choose the Footprint button. The Select Footprint dialog box appears.
4. In the Libraries group box, select LAYOUT.LLB. Use the Add button, if necessary, to add this library to the list of available libraries. (LAYOUT.LLB resides in the LIBRARY directory.)
5. In the Footprints group box, select a mounting hole (OrCAD provides three: MTHOLE1, MTHOLE2, and MTHOLE3). Choose the OK button to close the Select Footprint dialog box.
6. Select the Not in Netlist option, then choose the OK button to close the Add Component dialog box. The mounting hole attaches to your cursor.
7. Place the mounting hole by clicking the left mouse button.

Note: To have a mounting hole thermal into the plane layer, attach it to the net that is shorted to the plane layer. You can do this after placement.

Note: If you don't want a pad on the top, bottom, and inner layers, but need clearance on the plane layers, place pads that are 1 mil in diameter on the top, bottom, and inner layers. These 1 mil pads will be seen by SmartRoute and avoided, and will be drilled out when drill holes are drilled through the board. For the plane layers, you need to define pads that are 15 mils larger than the drill hole, to provide adequate clearance from the drill. Pad size on plane layers is used to define clearance. Plane layers are represented in the inverse, in Layout.

Defining the layer stack

Note: For instructions on how to copy a padstack layer to a newly defined layer, see Copying padstack layers.

Routing and documentation layers are defined in the Layers spreadsheet. Using the spreadsheet, you can define the number of routing layers that will be used for the board.

If you plan to have a board with four routing layers (TOP, BOTTOM, INNER1, and INNER2) and two plane layers (POWER, GROUND), then you need to define the layers in a technology template (.TCH) or a board template (.TPL).

Note: It is better to have too many routing or plane layers defined than too few (if you're unsure of the number you will need) before reading in a netlist, because you can decrease the number of the layers later, by designating them as unused.

To define layers for routing

1 Choose the spreadsheet toolbar button, then choose Layers. The Layers spreadsheet appears.

Note: Do not delete layers from the Layers spreadsheet. To disable a layer, double-click on it, then specify it as Unused Routing in the Edit Layer dialog box.

2 Review the type assignments for the routing layers and double-click in the Layer Name column of a layer you want to modify. The Edit Layer dialog box appears.

3 In the Layer Type group box, select the desired option (for example, to disable a layer for routing, select Unused Routing; to define an additional plane layer, select Plane Layer).

4 If you changed a routing layer to a plane layer, change the Layer LibName to PLANE.

5 Choose the OK button.

Related topics

Edit Layer dialog box

Defining global spacing values

Global spacing values set rules for spacing between the various objects on the board. You can define global spacing values for the board using the Edit Spacing dialog box, which is accessed from the Route Spacing spreadsheet.

You can save spacing requirements in a board template (.TPL).

Uniform spacing requirements per layer reduce processing time.

Note: To globally assign the same spacing to all layers, double-click in the Layer Name title cell in the Route Spacing spreadsheet. When the Edit Spacing dialog box appears, enter a value in the appropriate text box (for example, enter a value for Track to Track Spacing), then choose the OK button.

To define global spacing values

1 Choose the spreadsheet toolbar button, choose Strategy, then choose Route Spacing. The Route Spacing spreadsheet appears.

2 Double-click on the layer you want to modify. The Edit Spacing dialog box appears.

3 Set these options, then choose the OK button.

- **Track to Track Spacing**

  Tracks are defined as routed connections and copper obstacles, such as keepouts and place outlines. Track-to-track spacing specifies the minimum space required between tracks of different nets, and between tracks and obstacles of different nets.

- **Track to Via Spacing**

  Track-to-via (and obstacle-to-via) spacing specifies the minimum space required between vias and tracks of different nets.

- **Track to Pad Spacing**

  Track-to-pad (and obstacle-to-pad) spacing specifies the minimum space required between pads and tracks of different nets.

- **Via to Via Spacing**

  Specifies the minimum space required between vias of different nets.

- **Via to Pad Spacing**

  Specifies the minimum space required between pads and vias of the same net (as well as different nets, which is the usual case). For instance, to keep a distance of 25 mils between your SMT pads and the fanout vias connected to the pads, set Via to Pad Spacing to 25.

- **Pad to Pad Spacing**

  Specifies the minimum space required between pads of different nets.

Defining padstacks

Padstacks define the pads of the footprint and must be defined before assigning them to the footprint. They possess properties on each layer of the board, such as shape and size.

If you are using the standard Layout footprint libraries, or if you have made your own footprints using Layout standards, you have used padstacks T1 through T7 to create most of the standard through-hole components in your library.

Note: Don't name your custom padstacks using the names T1 through T7, because they will be overwritten by technology template padstacks whenever you load a technology template.

The use of each padstack definition is defined as follows:

T1:	Round IC pads
T2:	Square IC pads
T3:	Round discrete pads
T4:	Square discrete pads
T5:	Round connector pads
T6:	Square connector pads
T7:	Via SMT stringer pads

You can define new padstacks by copying and editing existing padstacks in the Padstacks spreadsheet. Then, you can assign them to footprints or footprint pins. After you create new padstacks, you can save them in a board template (.TPL) for use with future boards.

Note: For information on assigning padstacks to footprints or footprint pins, and on editing padstacks, see Creating and editing footprints.

Note: Be sure to define through-hole padstacks on all layers, including unused layers. Otherwise, you may unintentionally create blind or buried vias. Surface-mount pads are not defined on internal layers.

To create a new padstack

1. Choose the spreadsheet toolbar button, then choose Padstacks. The Padstacks spreadsheet appears.
2. Select a padstack that is similar to the one you want to create, then choose New from the pop-up menu. A new padstack is added at the bottom of the spreadsheet.
3. Select the new padstack and choose Properties from the pop-up menu.
4. Type a new name for the padstack in the Padstack text box, edit the other options to change the size or shape as desired, then choose the OK button.
5. Make specific layer definitions for the padstack for the drill and plane layers.

   To add a drill hole to the padstack, define a round padstack on the drill layer. Copper clearance around the drill hole is created by a round padstack on each plane layer that is larger than the drill size.

Defining vias

Layout provides one defined via and fifteen undefined vias. You define additional vias in the Edit Padstack dialog box (from the Padstacks spreadsheet) to make them available for routing.

Defining an unused via

To define an unused via

1. Display the Padstacks spreadsheet. Scroll down to the first via whose pad shape is entirely undefined, and select that via.
2. From the pop-up menu, choose the Properties command to display the Edit Padstack dialog box.

3 Define the via to suit your needs and choose the OK button.

4 From the Options menu, choose the Route Settings command to display the Route Settings dialog box.

5 Select the Use all via types option and choose the OK button. The new via is now available for routing.

Note: If you don't select the Use All Via Types option in the Route Settings dialog box, you must specifically assign vias to nets that need their via types restricted. Otherwise, the router chooses what it considers the "best" via, using its standard criteria: the layer(s) the via is defined on and its size compared to track size.

Related topics

Edit Padstack dialog box

Route Settings dialog box

Padstacks spreadsheet

Assigning a via to a net

To assign a specific via to a particular net

1 If the via is not yet defined, follow the steps above to create it.

2 In the Nets spreadsheet, select the particular net.

Note: Selecting a via for a particular net does not prohibit any other net from using that via. The assignments made in the Assign Via dialog box simply override, for selected nets, the Use All Via Types option set in the Route Settings dialog box (from the Options menu, choose Route Settings). Therefore, you can select the Use All Via Types option and still assign specific vias to specific nets using the Assign Via dialog box.

3 From the pop-up menu choose the Assign Via per Net command. The Assign Via dialog box.

4 Select the specific via and choose the OK button.

Note: You don't have to select the Use All Via Types option in the Route Settings dialog box to assign a via to a particular net.

Related topics

Nets spreadsheet

Assign Via dialog box

Changing via definitions

To change the definition of a via

1 Select one of the manual routing tools, then select the via in question.

2 From the Edit menu, choose the Change Via command. The Via Selection dialog box, with all the defined vias, presents itself.

3 Select the appropriate via and choose the OK button.

For information on changing the definition of a via, see Changing vias on page 418.

Related topics

Via Selection dialog box

Placing a via

To place a via

1 Choose one of the routing toolbar buttons.

2 Begin routing the net on which you want to place a via.

3 Click the left mouse button to place a vertex (a corner).

4 From the pop-up menu, choose Add Via or from the pop-up menu, choose Add Free Via.

Related topics

Generating test points

Viewing drill holes

You can view the drill holes on the board in the design window.

To view drill holes

1 In the left column of the Color spreadsheet check for the presence of Default DRILL.

2 If Default DRILL is present, proceed to step 4, below.

or

If it is not present, click the right mouse button in the Color spreadsheet window, then choose the New command from the pop-up menu to display the Add Color Rule dialog box.

3 In the Layer text box, remove the minus sign and enter DRILL, then choose the OK button. The Color spreadsheet is visible again.

4 Double-click on Default DRILL to display the Color dialog box.

5 Select a distinctive color, verify that the Invisible option is not selected, then choose the OK button.

6 Close the Color spreadsheet window.

7 From the Layer drop-down list box on the toolbar, choose the Drill layer (DRL). The drill holes display in the selected color.

Related topics

Color spreadsheet

Add Color Rule dialog box

Color dialog box

Drills spreadsheet

Viewing the drill drawing and drill chart

Exporting and importing ASCII files

Making a ratsnest invisible

Defining free vias

You can add free vias to your design for special purposes, such as zero-length fanouts of ball grid array (BGA) components and the “stitching” of plane layers.

Free vias (denoted by the letters FV) are ignored by Layout’s board cleanup routines, so you can place them on your board and have them stay there, as long as they are attached to a net. They are preserved through AutoECO, unless the net or routed track they are connected to is entirely deleted or removed from the board.

Layout regards free vias as stand-alone components: you can shove them, place them in isolation (free of tracks), or connect them to multiple tracks on the same net.

To define a free via

1. From the Tool menu, point to Via and choose New. The Add Free Via dialog box appears.
2. Select the padstack name you want to assign to the free via.
3. Select the net you want the free via attached to.
4. Determine the remaining option settings and click OK.

Setting net properties

Net properties affect manual routing, autorouting, and autoplacement. Most of the net data used in Layout is established at the schematic level using net properties. However, these rules can be enhanced or modified at any time during the design process.

Net data can be viewed and accessed in the Nets spreadsheet.

Related topics

Nets spreadsheet

Opening and editing nets

To open the Nets spreadsheet

1. Choose the spreadsheet toolbar button, then choose Nets. The Nets spreadsheet appears.

To edit net properties

1. In the Nets spreadsheet, double-click on a net. The Edit Net dialog box appears.
2. Edit the options in the dialog box as desired, then choose the OK button.

To find a net in the spreadsheet

1. In the Nets spreadsheet, choose Select Any from the pop-up menu. The Net Selection Criteria dialog box appears.
2. Enter the name of a net you are looking for, then choose the OK button. Layout highlights the net in the Nets spreadsheet and highlights the net on the board.

Note: If you select a net, then bring up the Nets spreadsheet, the selected net's row is highlighted in the Nets spreadsheet.

Related topics

Nets spreadsheet

Enabling layers for routing

In the Layers Enabled for Routing dialog box, you can specify on which layers a particular net can be routed. That is, you control which layers are enabled for routing on a per-net basis.

This option is valuable for nets that can only be routed on certain layers. The autorouter will not put a particular track on a layer unless the layer is enabled for routing for that net. An error occurs if you try to manually route a track on a layer that is not enabled for routing in the Layers Enabled for Routing dialog box.

To enable or disable layers for routing

1. In the Nets spreadsheet, select a net, then choose Properties from the pop-up menu. The Edit Net dialog box appears.

   Note: For instructions on enabling and disabling power and ground, see Routing the board.

2. Choose the Net Layers button. The Layers Enabled for Routing dialog box appears.

3. Select the layers on which you want to route the selected net, then choose the OK button.

Setting net widths by layer

Using the Net Widths By Layer dialog box, you can set a specific track width for each layer for each net. This feature is especially useful for impedance-controlled boards. If the width

of a net varies from its value as set in this dialog box, the design rule check flags it as an error.

After you set a net width using the Net Widths By Layer dialog box, you can change the width of the net later using the Force Width by Layer command (from the pop-up menu).

To set net widths by layer

1. In the Nets spreadsheet, select a net, then choose Properties from the pop-up menu. The Edit Net dialog box appears.
2. Choose the Width By Layer button. The Net Widths By Layer dialog box appears.
3. Edit the values as desired, then choose the OK button.

Setting reconnection order

Using the Reconnection Type dialog box, you can edit the reconnection rules for each type of reconnection allowed by Layout, and control the reconnection order.

To set the reconnection order

1. In the Nets spreadsheet, select a net, then choose Properties from the pop-up menu. The Edit Net dialog box appears.
2. Choose the Net Reconn button. The Reconnection Type dialog box appears.
3. Select a reconnection type for the net from the following options, then choose the OK button.

None

Maintains the existing net order.

Horizontal

Tells the router to seek primarily horizontal paths for each connection within a net. This option is generally used for power (VCC) and ground (GND).

High speed

Note: While routing, if you press the ALT key and click the left mouse button on a track, you can begin a new track on another track of the same net, which is known as T-routing.

Prohibits T-routing and tells the router to daisy-chain the connections in the net from the source to the load(s), and then to the terminator. This option is used for high speed nets, and is often used in conjunction with disabling share on critical nets.

Note: The source, loads, and terminators are set in the Packages spreadsheet. You must assign source and terminator pins in the Package Edit dialog box in order to use High speed for automatic ECL routing. Without these assignments, the router will daisy-chain the tracks, but will use an arbitrary source and terminator.

Vertical

Tells the router to seek primarily vertical paths for each connection within a net. This option is generally used for power (VCC) and ground (GND).

Std. Orthog.

Tells the router to seek the easiest path between any two points within a net. This is usually the shortest distance, but the option has a predisposition for horizontal or vertical routes where possible. This is the default option, and should be used for all routing of standard digital signals.

No Dyn. Reconn

By default, Layout uses dynamic reconnect, which is a method of calculating where the closest pin belonging to the same net you're routing is, then redrawing the ratsnest line to connect to the closest pin. The No Dyn Reconn option disables dynamic reconnect, with the result that you don't have to wait for Layout's ratsnest calculations and redrawing.

Because of this, selecting No Dyn Reconn is especially useful when routing large nets. Note that No Dyn Reconn is not available for use with the None or High speed types of reconnection, because they must maintain their connection orders.

Setting net spacing by layer

Using the Net Spacing By Layer dialog box, you can set the spacing per layer for each net so that you can precisely control the distance between any net and its neighbor. This applies to track-to-track spacing only, so that you can route critical signals between pins using the normal pad-to-track spacing.

The router always uses the largest spacing criteria that applies. Therefore, if the net-to-net spacing is 8 mils, but the global track-to-track spacing is 12 mils, the tracks remain 12 mils apart. This rule also applies to nets with different spacing.

The design rule check issues an error message if the specified minimum is violated.

To set net spacing per layer

1. In the Nets spreadsheet, select a net, then choose Properties from the pop-up menu. The Edit Net dialog box appears.
2. Choose the Net Spacing button. The Net Spacing By Layer dialog box appears.
3. Set the spacing for each layer for the selected net, then choose the OK button.

Creating and editing obstacles

6

Layout uses obstacles to restrict where components and tracks can be placed on a board. The most common types of obstacles are:

- Board outlines
- Copper pour
- Insertion outlines
- Place outlines

You can use IntelliCAD to create board outlines, keepins and keepouts, and similar objects. For information on IntelliCAD, see the IntelliCAD appendix

You can use the obstacle tool to create, edit, and place obstacles on your board. You can use the Edit Obstacle dialog box to choose the type of obstacle you want to create, and to set properties for the obstacle, such as size, target layer, and net attachment. Obstacles are used on the board and in the footprint library.

Because Layout remembers the physical properties of the last obstacle you created, you can easily create one or more similar obstacles in succession, including net and component properties, but of varying sizes.

Creating obstacles

When creating an obstacle, you first define it, then draw it.

To create an obstacle

1. Choose the obstacle toolbar button.
2. From the View menu, choose Zoom Out and click on the screen until you can view the entire board. Press ESC to exit zoom mode.
3. Press INSERT.
4. The cursor changes from a large cross (idle mode) to a small cross (active mode). Locate the point at which you want to start drawing the outline. There are three ways to move the cursor to this point: you can move the mouse, you can use the arrow keys, or you can press the TAB key to go to the desired X, Y coordinates. Click the left mouse button once on the screen. You will begin drawing from this point.

   Note: To place an obstacle at exact coordinates or coordinates that are off-grid, choose the find toolbar button. In the Find coordinate or Component Name dialog box, enter the coordinates (X, Y) at which you want to place the first corner and choose the OK button. Repeat for the other three corners.

   Note: If you are using a fine detail grid, use the mouse to approach the starting point, and then use the arrow keys to position the cursor. Once you are at the starting location, click the left mouse button to start drawing the obstacle, or press the spacebar to eliminate accidental mouse movement.
5. Double-click the left mouse button. The Edit Obstacle dialog box appears.

   Note: The Edit Obstacle dialog box includes special options based on the type of obstacle you are creating. For a detailed description of each option, the Edit Obstacle dialog box.

6 In the Obstacle Name text box, enter a name or leave the default number.

7 From the Obstacle Type drop-down list, select the type of obstacle you want to create.

8 In the Group, Height, Width text box, enter a value. The appropriate option is enabled, depending on the type of obstacle you are creating.

9 From the Obstacle Layer drop-down list, select the layer on which you want to place the obstacle.

10 From the Net Attachment drop-down list, select a net to attach or leave the default dash, then choose the OK button.

11 Move from the starting coordinates to the desired location of the first corner. Click the left mouse button or press the spacebar to insert the first corner. Move to the desired location of the next corner. Click the left mouse button or press the spacebar to insert the second corner.

Note: When creating an obstacle that is a line (free copper, detail, and so on), drag the cursor to draw the line, click the left mouse button to stop drawing, then choose End Command from the pop-up menu.

12 When you complete the final corner, choose Finish from the pop-up menu. Layout automatically completes the obstacle.

Selecting obstacles

To select an entire obstacle

1 Choose the obstacle toolbar button.

2 Press the CTRL key and select an obstacle or press and hold the left mouse button while dragging across a portion of an obstacle.
You can select multiple obstacles by pressing the CTRL key and clicking on the additional obstacles that you want to select. Selected obstacles are highlighted.

To select a segment of an obstacle

1. Choose the obstacle toolbar button.
2. Click on a segment with the left mouse button.

Editing obstacles

Use the Edit Obstacle dialog box to edit obstacles. Using the dialog box, you can choose the obstacle type and set physical properties, such as width, layer, and hatch pattern. You can also specify attachments for the obstacle, including footprints, components, pins, and net attachments.

Note: You can use the Edit Obstacle dialog box to set the properties for an obstacle before creating it, as described in Creating obstacles in this chapter.

To edit an obstacle

1. Choose the obstacle toolbar button.
2. Press the CTRL key and select an obstacle.
3. From the pop-up menu, choose Properties. The Edit Obstacle dialog box appears.
4. Edit the options as desired, then choose the OK button.

Copying obstacles

You can copy existing obstacles and place them on any layer.

To copy obstacles

1. Choose the obstacle toolbar button.
2. Press the CTRL key and select an obstacle.
3. From the pop-up menu, choose Copy.
4. Drag the copy to a desired location, then click the left mouse button to place it.

To copy obstacles to other layers

1. Follow the four-step procedure in *To copy obstacles* above.
2. Press the CTRL key and select the obstacle.
3. From the View menu, choose Select Layer. The Select Layer dialog box appears.
4. Select the target layer from the drop-down list, then choose the OK button.
5. Click the left mouse button to place the obstacle on the target layer. The obstacle's color changes to the color of the target layer.

Moving obstacles

To move an obstacle

1. Choose the obstacle toolbar button.
2. Press the CTRL key and select an obstacle.
3. Pressing the left mouse button, drag the obstacle to the new location.

To move an obstacle to another layer

1. Choose the obstacle toolbar button.
2. Press the CTRL key and select an obstacle.
3. From the View menu, choose Select Layer. The Select Layer dialog box appears.
4. Select the target layer from the drop-down list, then choose the OK button.
5. Click the left mouse button to place the obstacle on the target layer. The obstacle's color changes to the color of the target layer.

To move a segment of an obstacle

1. Choose the Obstacle Tool from the toolbar.
2. Select the obstacle.
3. Press s.
4. Move the pointer and the attached obstacle to the new location.
5. Click the left mouse button to place the obstacle.

Rotating obstacles

You can rotate obstacles using the Rotate command. However, you must first set the increment of rotation in the System Settings dialog box. Layout supports any rotation value.

To rotate an obstacle

1. From the Options menu, choose System Settings. The System Settings dialog box appears.
2. In the Increment text box, enter the value (in degrees) by which you want to rotate the obstacle, then choose the OK button.
3. Choose the obstacle toolbar button.
4. Press the CTRL key and select an obstacle.
5. From the pop-up menu, choose Rotate.

Mirroring obstacles

Mirroring rotates an obstacle around the X-axis. It does not automatically change the layer to the opposite layer. You can change the layer manually by typing the appropriate layer number while the obstacle is still attached to your pointer. If you want the obstacle to automatically appear on the opposite layer, press O. The opposite layer is defined in the Layers spreadsheet.

To mirror an obstacle

1. Choose the obstacle toolbar button.
2. Press the CTRL key and select an obstacle.
3. From the pop-up menu, choose Mirror. Layout mirrors the obstacle on the current layer
   or
   From the pop-up menu, choose Opposite. Layout mirrors the obstacle on the opposite layer.

Related topics

Mirrored footprints in Capture and Layout

Exchanging the ends of obstacles

After you select a linear obstacle, you can use the Exchange Ends command to move the pointer to the end opposite the current selection.

To move the pointer to the opposite end of a linear obstacle

1. Choose the obstacle toolbar button.
2. Select a segment or end of the linear obstacle by clicking on it with the left mouse button.
3. From the pop-up menu, choose Exchange Ends.

Moving segments

When you select a segment on an obstacle and attempt to move it, a vertex (a corner) is created. Use the Segment command to move entire segments without forming vertices. You can use this command to make obstacles larger or smaller.

To move a segment

1. Choose the obstacle toolbar button.
2. Select a segment, or side, of the obstacle.
3. From the pop-up menu, choose Segment.
4. Drag the segment to a new location. The segment moves, allowing you to extend or compress the entire side of the obstacle.

Creating circular obstacles

You can create circular shapes using the Arc command.

To create a circular obstacle

1. Choose the obstacle toolbar button.
2. From the pop-up menu, choose New.
3. Double-click at the point on the screen that you want to designate as the center of the arc. The Edit Obstacle dialog box appears.
4. From the Obstacle Type drop-down list, choose an obstacle type, edit other options in the dialog box as desired, then choose the OK button.
5. From the pop-up menu, choose Arc.
6. Drag the cursor to begin creating a circle.
7. Click the left mouse button to stop drawing.

Note: If you select an obstacle segment and type the letter A, an arc forms. Drag the arc to the desired coordinates and click the left mouse button to stop drawing.

Deleting obstacles

To delete an obstacle

1 Choose the Obstacle Tool toolbar button.

2 Select the obstacle with a selection rectangle.

3 Press the DELETE key.

OR

1 Choose the Obstacle Tool toolbar button.

2 Select the obstacle with a selection rectangle.

3 From the pop-up menu, choose the Delete command.

If you select the obstacle and use the pop-up menu to delete it, only one segment at a time is deleted.

Related topics

Select Tool command (Obstacle)

Creating alignment targets

To maintain accurate artwork registration through all layers, you can place alignment targets when you create the board outline, or at any later step in the process.

You can create your own alignment target shape as an obstacle of type Free Copper.

Another alignment target option is to use the Layout footprint, the Moiré, found in the LAYOUT.LLB library. The Moiré footprint is present on all the standard alignment target layers. If you want alignment targets on a different set of layers, you can specify one board layer (an unused routing or documentation layer) as the Moiré layer, place the footprint on that layer, then create your plot files with the Moiré layer, as well as the subject layer, set to visible.

To place an alignment target

1. Choose the Component tool, then choose New from the pop-up menu. The Add Component dialog box displays.
2. In the Component flags group box, verify that Route Enabled is not selected, that Not in Netlist is selected, and that either Fixed or Locked, as appropriate, is selected.
3. Choose the Footprint button. The Select Footprint dialog box appears.
4. Select the LAYOUT.LLB library, choose the Moiré footprint, then choose the OK button.
5. Position your pointer at the alignment target location, then click the left mouse button.

Related topics

Add Component dialog box

Creating and editing text

7

You can use text to label packages and pins, create reference designators, or to add information such as manufacturing notes to the board.

Adding, copying, and deleting text uses many of the same techniques you use when working with obstacles.

Creating text

Use the Text Edit dialog box to create all of the text you need to label your board and library parts.

To create text

1 Choose the text toolbar button.

2 Press the INSERT key. The Text Edit dialog box appears.

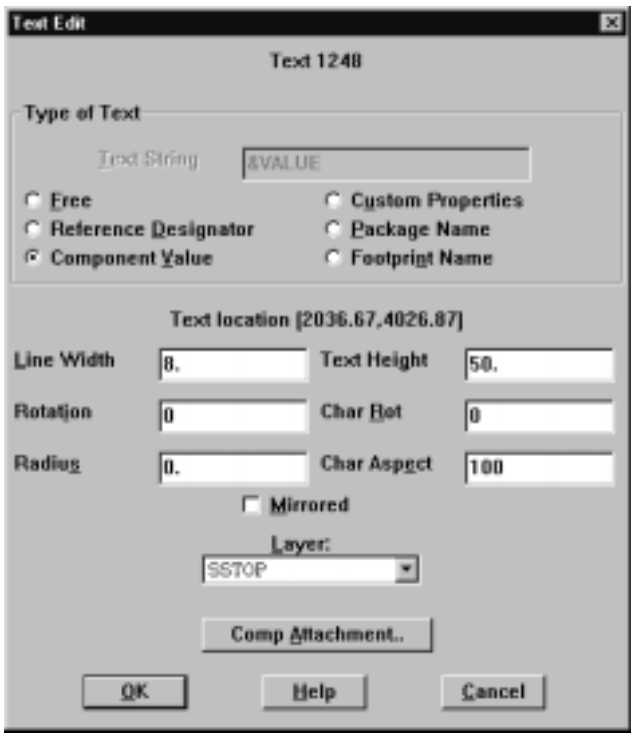


Figure 7-1 The Text Edit dialog box

3 From the Type of Text group box, select the type of text that you want to create.

4 If you select the Free option or the Custom Properties option, type a text string into the Text String text box. These options are described in the Text Edit dialog box description in this section.

5 Edit the Line Width, Rotation, Radius, Text Height, Char Rot (character rotation), and Char Aspect (character aspect) text boxes as desired. These options are described in the Text Edit dialog box description in this section.

6 Select the Mirrored option if you want the text to appear mirrored on the layer (useful for placing text on the bottom of the board).

7 Select the target layer from the Layer drop-down list.

8 If desired, choose the Comp Attachment button, select the Attach to Component option, supply the component's reference designator, then choose the OK button.

9 Choose the OK button to close the Text Edit dialog box.

10 Position the text on the screen and click the left mouse button to place it.

Text String

You must enter a text string if you choose the Free option or Custom Properties option in the Type of Text group box. Enter the text string as you want it to appear on the board.

If you select any of the other options from the Type of Text group box, the placeholder text appears in the text box. For example, if you are adding a reference designator to the footprint, it will display as *&Comp* in the Text String text box and in the library manager.

Note: The symbol & signifies a macro and should not be interpreted as a literal piece of text. The text string you see in the library manager, such as *&Comp*, is a placeholder that is replaced on the board by the actual name, value, or property as described and assigned by the schematic netlist.

Free

When you select this option, you can enter any text in the Text String text box, such as a serial number, to display on the board.

Reference Designator

The reference designator is supplied by the schematic netlist. The text string *&Comp* acts as a placeholder in the library. It is replaced by the appropriate reference designator when the footprint is attached to the component on the board.

Component Value

Select this option to display component values from the schematic netlist on the board.

The text string *&Value* acts as a placeholder in the library. It is replaced by the appropriate component value when the footprint is attached to a component on the board. For example, the component value of a resistor may be 10k. The placeholder *&Value* appears in the footprint editor, but after the footprint is attached to the resistor, the value 10k appears on the board.

Custom Properties

Select this option to display selected properties from the schematic netlist on the board. These properties can include part numbers and other details.

You must type the appropriate placeholder (as defined at the schematic level) in the Text String text box. For example, to display the part number on the footprint, type *&Partnumber* in the Text String text box. The text string *&Partnumber* acts as a placeholder in the library, until the footprint is attached to the component on the board. Then, *&Partnumber* is replaced by the actual part number of the component, as supplied by the schematic netlist.

Package Name

The package name is supplied by the schematic netlist and is used to describe the logical or internal characteristics of a component. The text string *&Pack* or *No Package* appears as a placeholder in the library, but it is replaced by the appropriate schematic netlist information when the footprint is attached to a component on the board.

Footprint Name

Select this option to display the name of the footprint on the board. If you choose this option, Layout prompts you to specify

a component attachment, and displays the Comp Attachment dialog box.

Text location

Displays the current coordinates of the text.

Line Width

Specifies, in characters, the width of the text line.

Rotation

Specifies, in degrees, the rotation of a text line.

Radius

Assigns a radius (circular shape) to a text string.

Text Height

Specifies text height.

Char Rot

Rotates individual characters.

Char Aspect

Assigns the width of the letters relative to the height.

Mirrored

Reflects the text on the mirror layer.

Layer

Specifies the layer on which the text is to display.

Comp Attachment

Displays the Comp Attachment dialog box, in which you can attach text to a component by supplying the component's reference designator.

Moving text

To move text

1. Choose the text toolbar button.
2. Click on the text with the left mouse button. It attaches to the cursor.
3. Move the mouse to position the text in the new location.
4. Click the left mouse button to place the text.

Deleting text

To delete text

1. Choose the text toolbar button.
2. Select the text by clicking on it with the left mouse button.
3. Press the DELETE key.

Design Reuse

8

When placing and routing a board file, it is common for parts of a board to be identical to previous board designs. For example, a power supply from a previous board might be identical to a current board file. If the previous and current boards share a common Capture schematic, and if that schematic is properly annotated, it is possible to apply design reuse and extract placement and routing information from the previous MAX file. This type of design reuse is called external design reuse.

Design reuse can also be applied to reuse a part of the current board. This type of reuse is called internal design reuse. An example of internal design reuse could be a multi-channel amplifier, where the placement and routing for each channel is easily reused.

Related topics

Placing and editing components

Routing the board

Design Reuse dialog box

Design Reuse command

Internal Design Reuse

Internal design reuse refers to using the placement and routing information of one part of a design to place and route another part of the same design. A Capture design with reuse of this type will typically contain multiple instances of a schematic that are referenced through a hierarchical object.

Note: A reuse schematic includes all sub schematics within that schematic, unless a sub schematic is also a reuse schematic.

When a Capture design containing internal design reuse is annotated, the Capture Annotation tool determines the part packaging and ensures that the part packaging is consistent across all reuse schematics.

Related topics

External Design Reuse

Partial Design Reuse

Creating the Capture design for internal reuse

Design reuse begins with and is controlled by the organization of the Capture schematic. Items to be reused are grouped at the schematic level in Capture. Everything contained within a reuse schematic is reused. So, to reuse placement and routing, the reuse schematic must be placed as a hierarchical block. The reuse schematic can be anywhere within the design hierarchy, and does not need to be placed in the root schematic.

To create an internal design reuse schematic:

1. Identify the part of your design that you want to reuse.
2. Create the reuse schematic. Only include the parts that you want to reuse. Gates should not be used from packages that are not contained within the design reuse

schematic. See Partial Design Reuse for more information about sharing packages.

3 Place a hierarchical block, and reference the reuse schematic.

a. From the Place menu, choose Hierarchical Block

b. Enter a name for the hierarchical block in the Reference text box.

c. In the Implementation Type list box, choose Schematic View as the implementation type,

d. Type the name of the schematic folder in the Implementation Name text box.

e. Since the reuse schematic is part of the current design, you do not need to specify a path to the schematic folder in the Path and filename text box.

f. Click OK.

g. Use the cursor to draw the boundaries of the hierarchical block on the schematic page. Capture creates the new hierarchical block and automatically places the hierarchical pins according to the ports that exist in the attached schematic.

Related topics

Annotating the design for internal reuse

Creating a netlist in Capture for Layout

Placing and routing the first instance of reuse

Applying internal design reuse to reuse targets

Annotating the design for internal reuse

Once the Capture design has been drawn, wired, and footprints assigned, it needs to be annotated properly before creating a netlist.

For design reuse to work properly, the following annotation requirements for a Capture design must be followed.

- Duplicate references are not allowed in the design.
- Multi-part packages devices in design reuse schematics should not be packaged with multi-part packages outside of the design reuse schematic.
- Multi-part package devices should have consistent packaging across reuse schematics. This includes reference numbering and part packaging information.

The Layout Reuse tab in Capture's Annotate dialog box follows the above rules.

Note: For exceptions to the design reuse rules, see Partial Design Reuse.

To annotate the Capture schematic for Layout design reuse:

1. Open the project file in Capture.
2. Highlight the design file in the project manager.
3. From the Tools menu, choose Annotate.
4. In the Annotate dialog box select the Layout Reuse tab.
5. In the Action group box, choose the appropriate annotate action. For more information about each action, see the Layout Reuse Tab (Capture Annotate dialog box) topic.
6. In the Physical Packaging group box enter the properties that must match for Capture to group the parts into a single package. The Value and Source Library properties are used as the default property string, but you can use any combination you like.
7. In the Select schematic(s) to mark for reuse: box, check all schematics that need to be marked for reuse. By selecting a schematic for reuse, it informs the annotation tool to treat this schematic (and all of its sub schematics) independently from the rest of the design. The reuse schematic is annotated following the required rules for Layout design reuse.

8 Click OK to annotate the schematic.

9 After the annotation is complete, check the Capture Session Log to see if there were any annotation errors. If there are errors after an incremental reference update, you may need to unconditionally annotate the schematic.

Note: If you are implementing partial design reuse, see Partial Design Reuse for acceptable warnings.

Related topics

Creating the Capture design for internal reuse

Creating a netlist in Capture for Layout

Placing and routing the first instance of reuse

Applying internal design reuse to reuse targets

Creating a netlist in Capture for Layout

Note: Due to additions in the netlist format, you must use Capture 10.0 or newer to create the netlist. Older versions of Capture do not support Layout design reuse.

To create a netlist for Layout design reuse:

1 Highlight the design file in the Capture project manager.

2 From the Tools menu choose Create Netlist.

3 Select the Layout tab in the Create Netlist dialog box and create the Layout netlist as usual.

Related topics

Creating the Capture design for internal reuse

Annotating the design for internal reuse

Placing and routing the first instance of reuse

Applying internal design reuse to reuse targets

Placing and routing the first instance of reuse

Since this is internal design reuse, it is necessary to place and route the first instance of design reuse. The design reuse tool uses this first instance of design reuse to place and route the other instances of design reuse.

To place and route the first instance of reuse:

1. In the Layout session window, choose New from the File menu to create a new board.
2. In the AutoECO dialog box choose the appropriate settings to create the new board file and click Apply ECO. For more information, see the AutoECO dialog box help topic.
3. Locate all components in the first instance of design reuse. To make the task of locating all of the components in the jumble of components of a new board, try grouping the components together.
   To group and place components:
   - **a.** Click the View Spreadsheet toolbar button and choose Components.
   - **b.** In the Components spreadsheet, CTRL-Click each component in the first instance of design reuse.
   - **c.** Right-Click one of the highlighted components and choose Properties from the pop-up menu.
   - **d.** In the Edit Component dialog box, enter a number into the Group text box. This can be any number you choose.
   - **e.** Click OK.
   - **f.** From the Tool menu, point to Component and choose Select Filtered.

g. In the Component Selection Criteria dialog box, enter the Group number you created earlier into the Group text box and click OK.

h. All components of this group are now highlighted, and you can easily click and drag them to a new area to visually set them apart from the other components.

4 Place and route all of the components in the first instance of design reuse.

If you do gate swap or pin swap on a reuse block, you must do the same gate swap or pin swap on all instances of that reuse block, back annotate the changes to Capture, create the netlist for Layout in Capture, load the netlist into Layout, and then do design reuse.

Related topics

Creating the Capture design for internal reuse

Annotating the design for internal reuse

Creating a netlist in Capture for Layout

Applying internal design reuse to reuse targets

Applying internal design reuse to reuse targets

After the first instance of design reuse has been placed and routed, you are now ready to apply design reuse.

To apply design reuse:

1 From the Auto menu choose Design Reuse. The Design Reuse dialog box appears.

2 In the Select source schematic box, navigate to the instance that you have already placed and routed. Click to check the box by the source of the design reuse.

Note: Layout determines design reuse based upon the organization of the source schematic. You need to know the location of the first instance of reuse within that schematic.

3 If you wish to move all of the components and routing in the first instance of design reuse, check Reposition the source. When this option is selected the first instance placed will be the source.

4 In the Select target schematic(s) box all available targets of design reuse are listed. Check all instances that you want to place and click OK.

5 The first instance of design reuse is now attached to your cursor. Click the left mouse button to place it. If you selected more than one target for reuse, the next target is automatically attached to the cursor. Press the ESC key at any time to cancel the placement of the additional targets.

Note: Undo is not supported when using design reuse. You will not be able to undo any placement and routing applied with design reuse.

Related topics

Creating the Capture design for internal reuse

Annotating the design for internal reuse

Creating a netlist in Capture for Layout

Placing and routing the first instance of reuse

Internal design reuse example

Creating the Capture design for internal reuse

Note: The designs presented in this example do not create an actual product, but merely serve to demonstrate the concepts of internal design reuse with Capture and Layout.

The design in this example, INTERNAL.DSN is a simple complex hierarchical design. The design consists of a ROOT schematic, SCHEMATIC1 with two hierarchical blocks, Reuse1 and Reuse2 that reference the same internal

schematic, SCHEMATIC2. Each schematic consists of 3 NAND gates and a resistor.

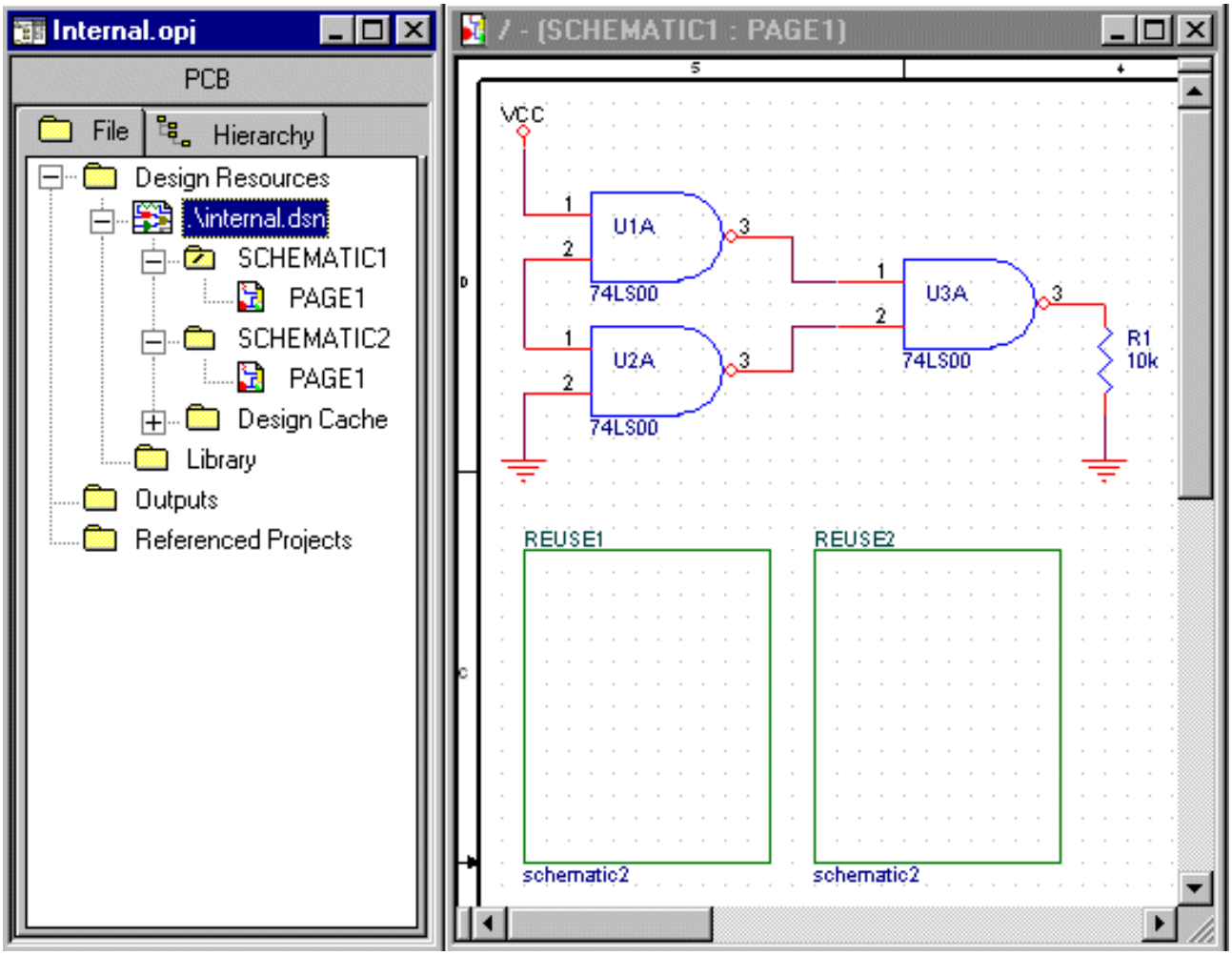


Figure 8-1 A design with internal design reuse

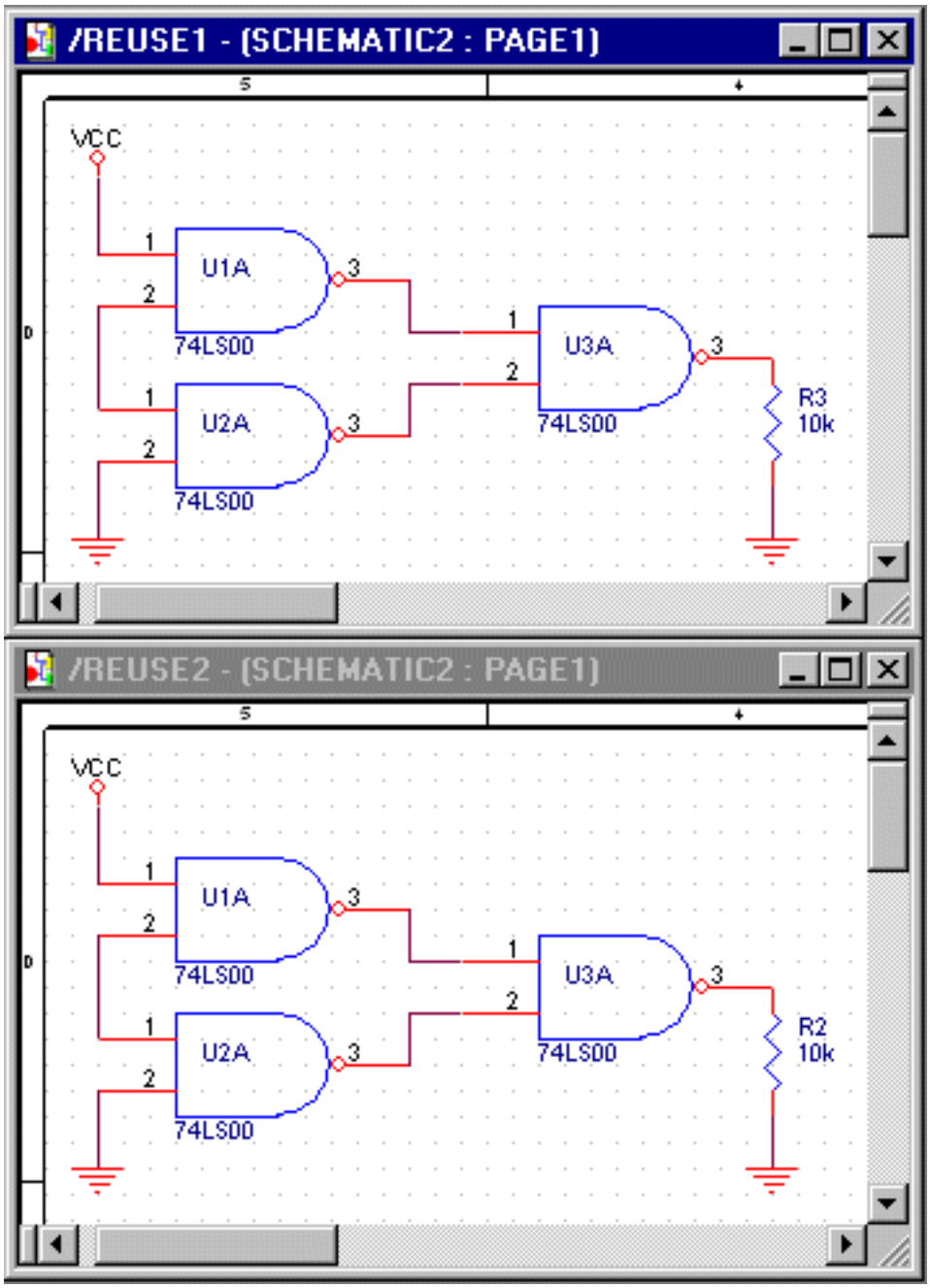


Figure 8-2 Schematics in the internal design reuse design

Before placing the hierarchical blocks in the root schematic page, first create the reuse schematic. In this example, the reuse schematic is SCHEMATIC2

Next, place a hierarchical block in the root schematic page. Reference the reuse schematic it in the hierarchical block properties.

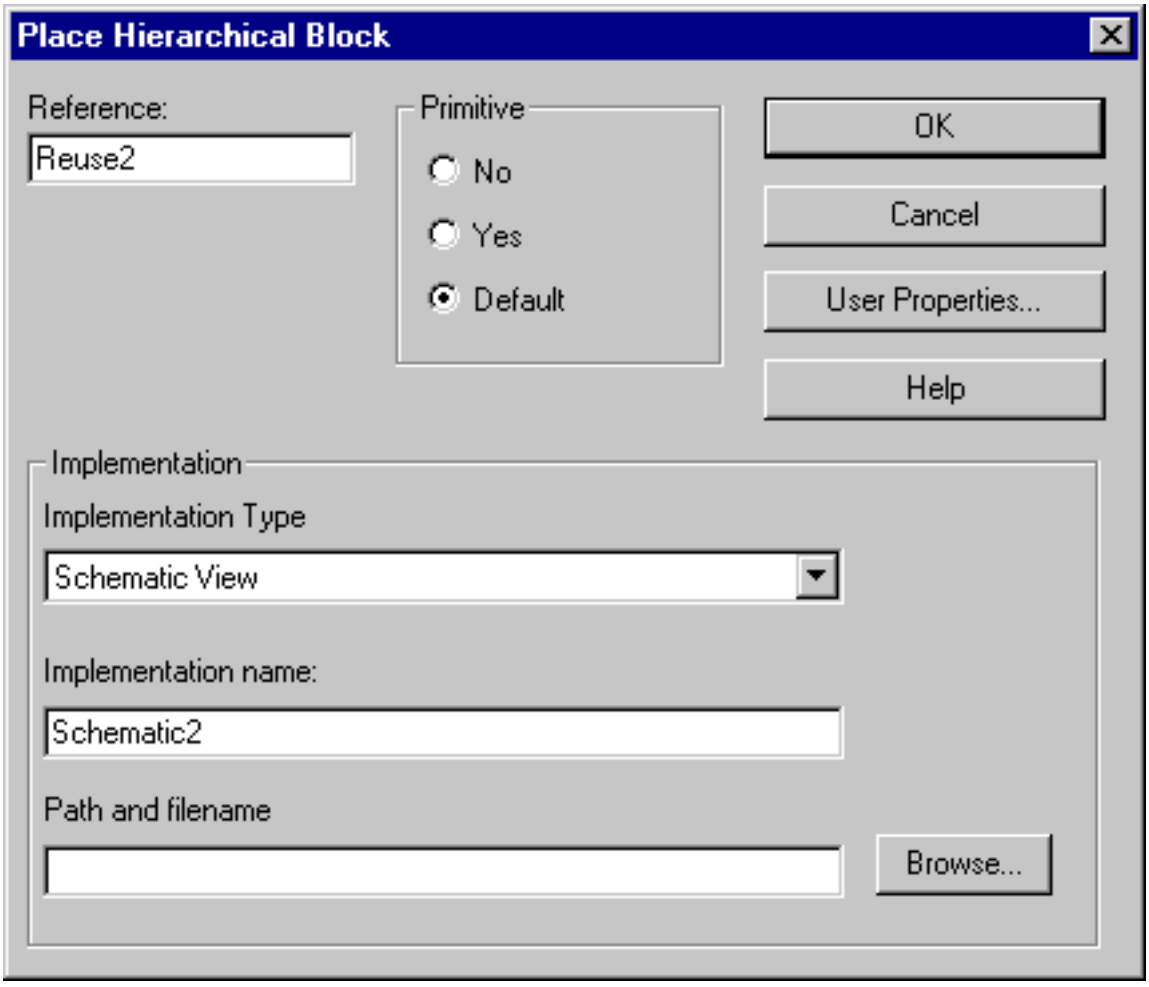


Figure 8-3 Placing a hierarchical block

In this example, the reference name of the hierarchical block is Reuse2. The hierarchical block references the internal schematic called SCHEMATIC2. Schematic View is selected as the Implementation Type.

Annotating the design for internal reuse

Once the Capture design has been drawn, wired, and footprints assigned, it needs to be annotated properly. The Capture design does not currently meet the annotation design requirements for Layout design reuse, specifically because it has duplicate references in the hierarchical blocks. So, it is necessary to unconditionally annotate this design for Layout Reuse.

To annotate this design for reuse, select the Layout Reuse tab in the Annotate dialog box.

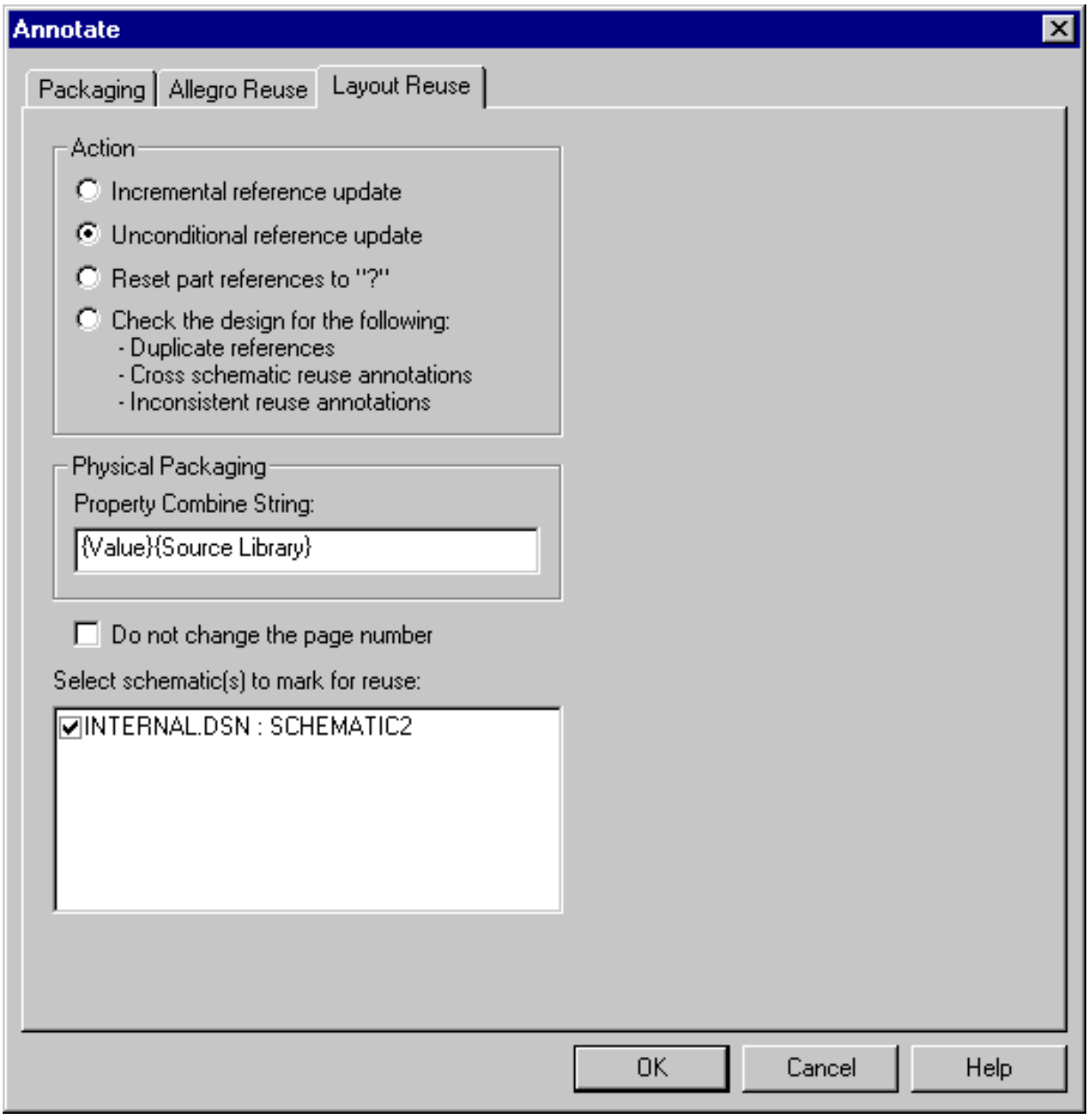


Figure 8-4 The Annotate dialog box in Capture

The above dialog shows the Layout Reuse tab configured to annotate this design properly. For this example, an Unconditional reference update has been chosen. Notice that

SCHEMATIC2 is selected as the reuse schematic in the Select schematic(s) to mark for reuse list. SCHEMATIC1 is not listed because it is the root schematic in the current design. By selecting SCHEMATIC2 for reuse, it informs the annotation tool to treat this schematic (and all of its sub schematics) independently from the rest of the design. The reuse schematic is then annotated following the required rules for Layout design reuse.

After the annotation is complete, the Capture Session Log indicates any errors. The following is the log file for the example design. The log indicates the annotation was successful, and that no annotation errors were found.

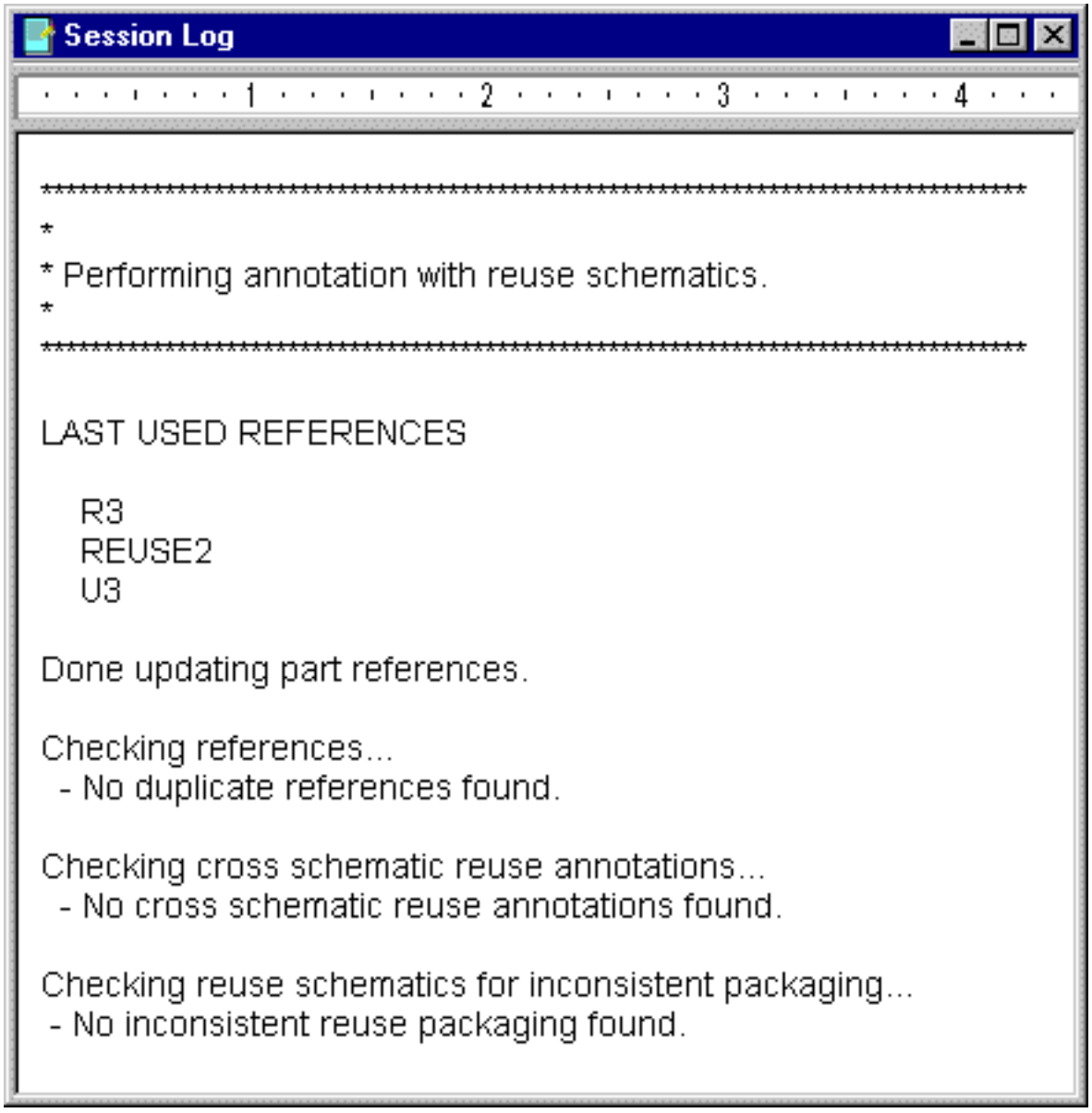


Figure 8-5 The Session Log in Capture

The following shows the example design properly annotated for internal design reuse.

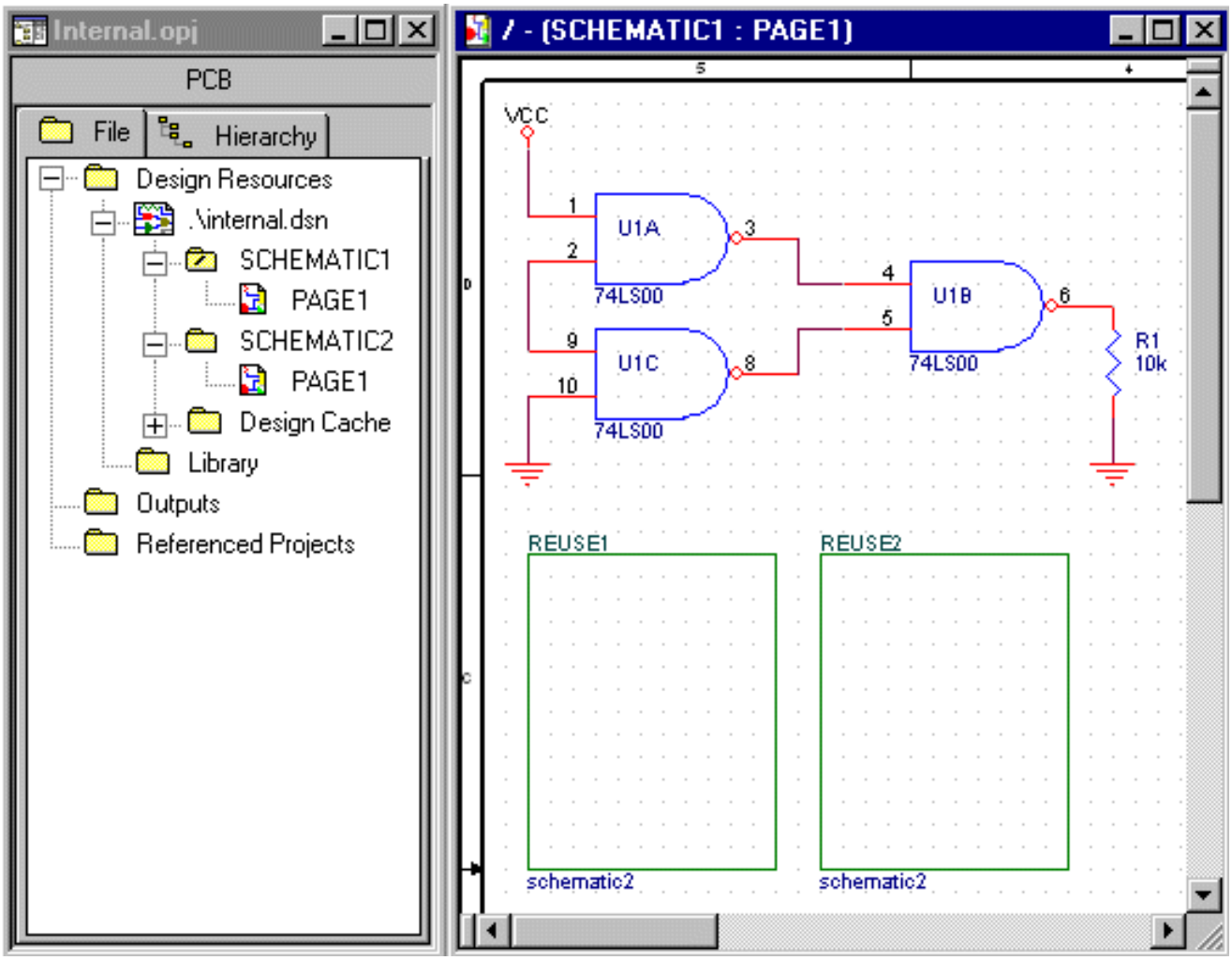


Figure 8-6 A design properly annotated for design reuse

Each reuse schematic (Reuse1 and Reuse2) has been packaged separately - from each other, and from the root schematic. Also note that in each reuse schematic, the corresponding parts have been assigned to the same package. It is only coincident that the roots schematic parts

have the same packaging as the reuse schematics because the root has the same number of NAND gates.

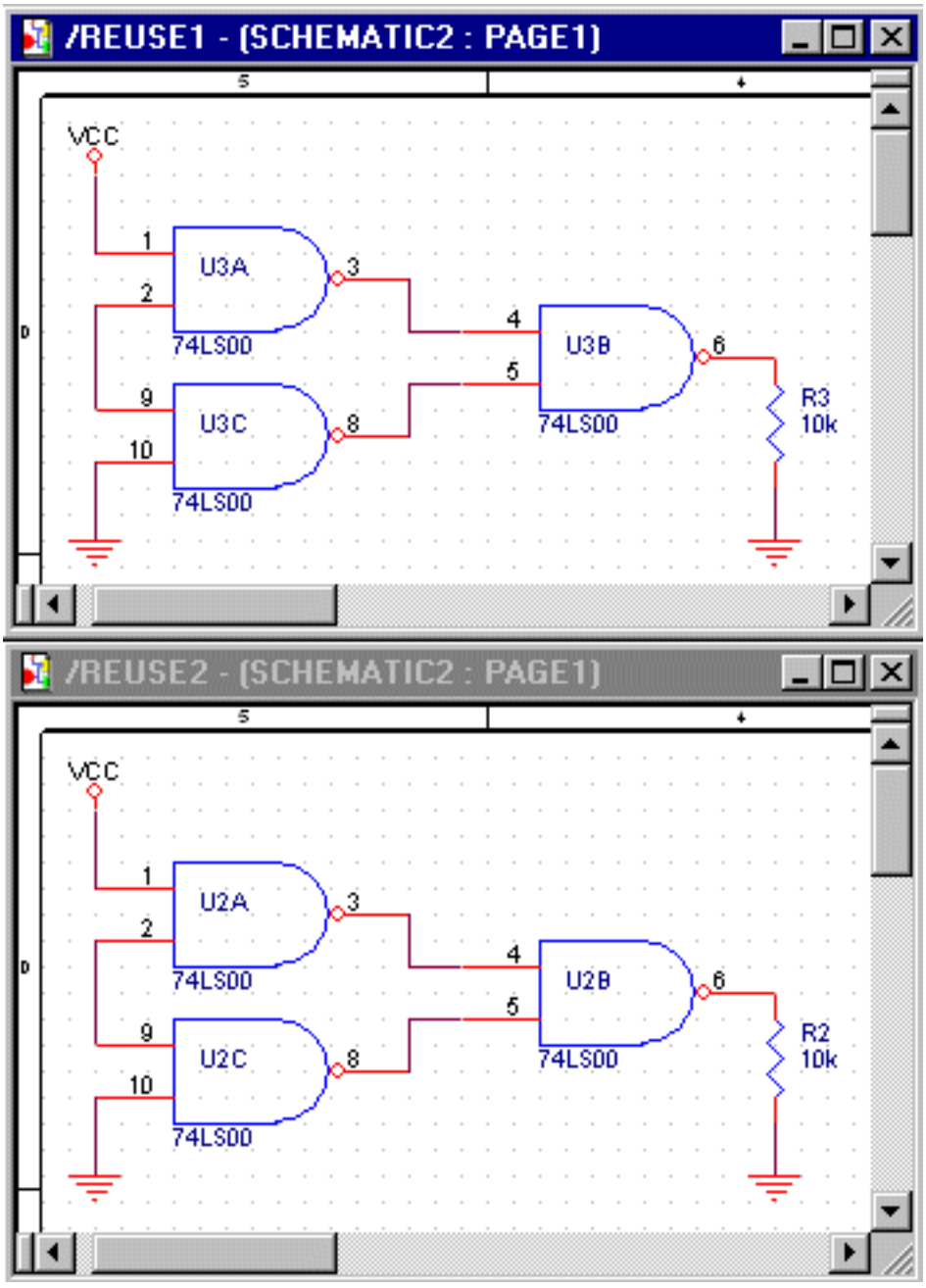


Figure 8-7 Schematics properly annotated for design reuse

This design now meets the Layout reuse annotation requirements for Layout design reuse. It is now possible to create the Layout netlist.

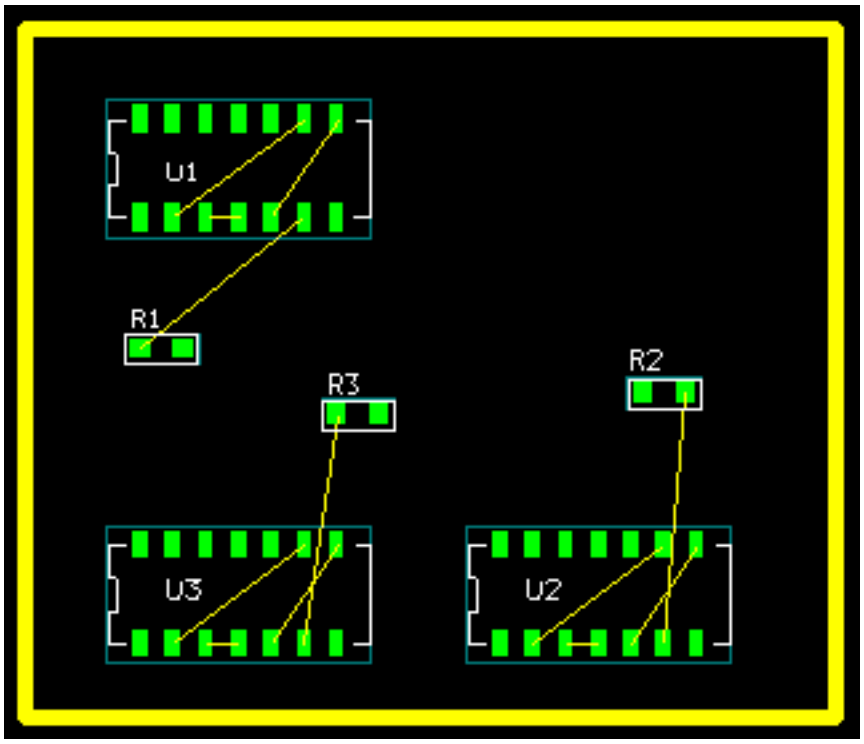
Creating a netlist in Capture for Layout

To create a netlist for Layout design reuse, the Layout netlist is created as it would be for any other design.

Note: Capture 10.0 or newer must be used to create the netlist. Older versions of Capture do not support Layout design reuse.

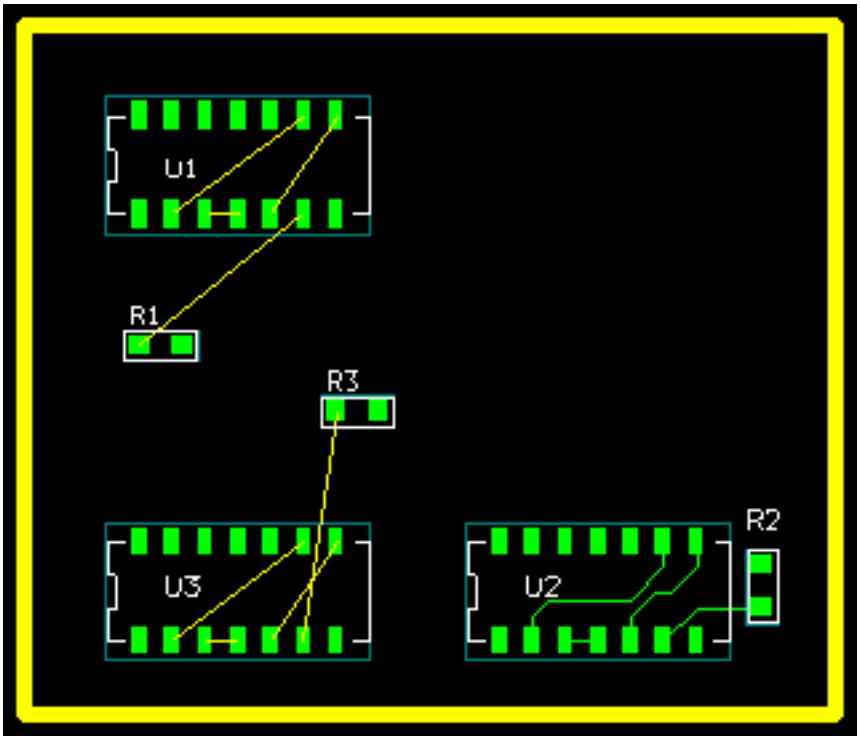
Placing and routing the first instance of reuse

The netlist is loaded into Layout and processed with the AutoECO utility. A board outline large enough to hold all of the parts is created, and then all of the components are block dragged into the outline. Since the components U1 and R1 are not reused or the source of reuse, they are placed to the side out of the way.



Note: The VCC and GND nets, and the DEFAULT ASYTOP layer have been disabled for clarity.

Since this design contains internal design reuse, there is no current placement and routing information to use as the source. The source must be manually placed and routed. It is then possible to use the source to apply design reuse to the other design reuse targets. In this example, there are two sources from which to choose. U2 and R2 have been chosen as the source, and are manually placed and routed.



It is now possible to apply internal design reuse to place and route components U3 and R3.

Important

If you do gate swap or pin swap on a reuse block, you must do the same gate swap or pin swap on all instances of that reuse block, back annotate the changes to Capture, create the netlist for Layout in Capture, load the netlist into Layout, and then do design reuse.

Applying internal design reuse to reuse targets

Applying internal design reuse copies the component placement and routing information to the target components.

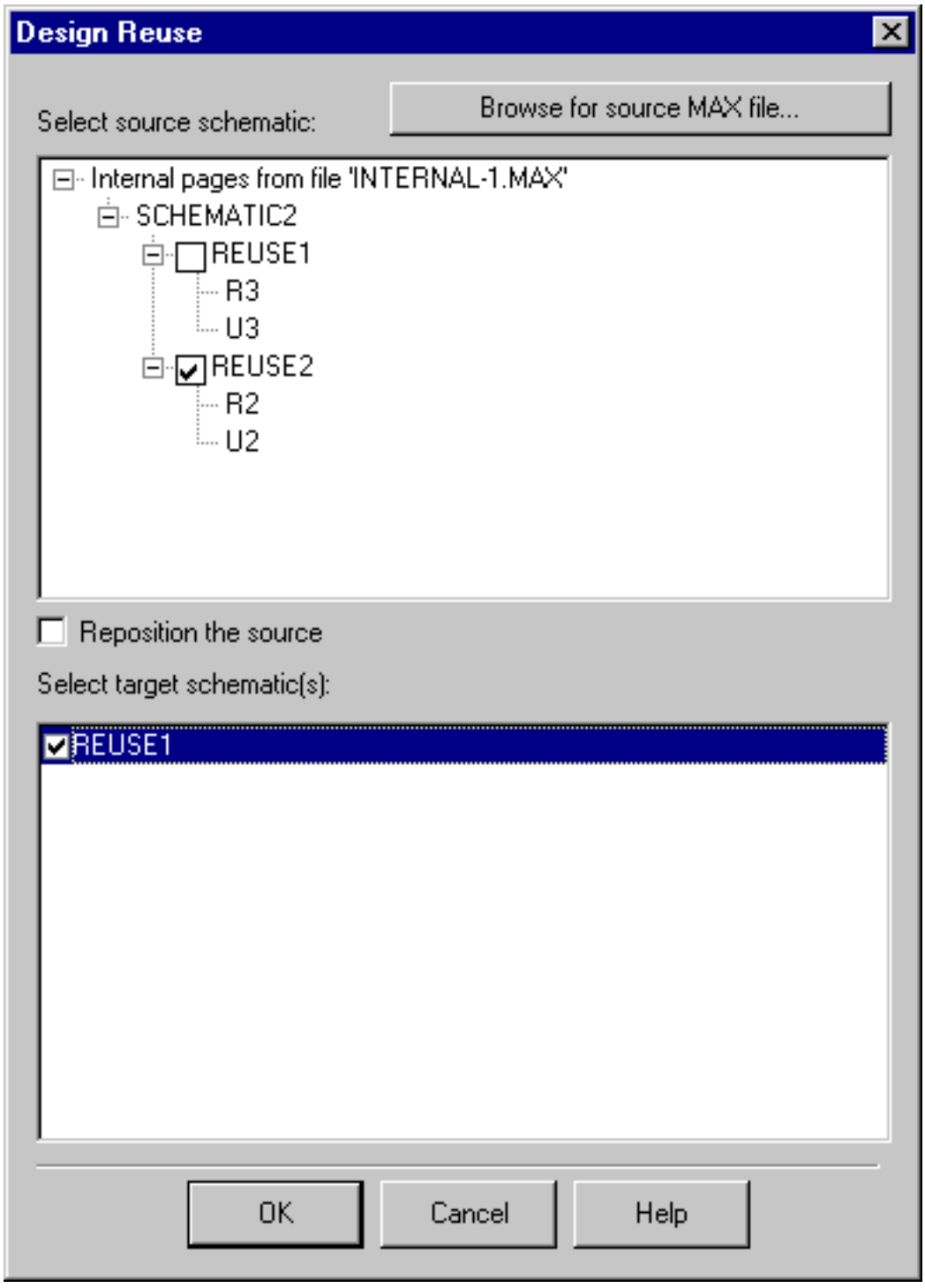


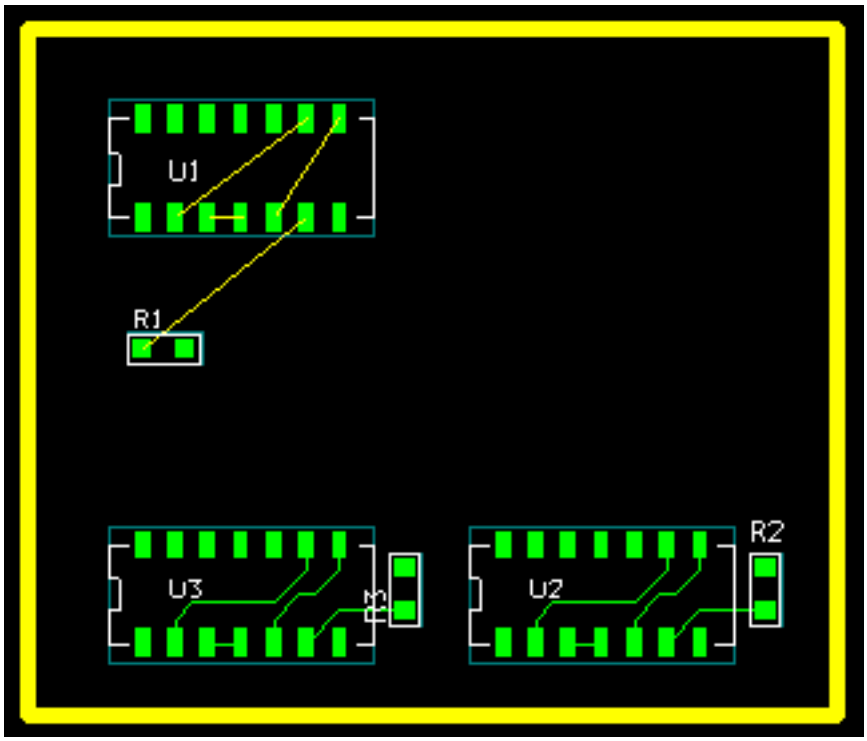
Figure 8-8 The Layout Design Reuse dialog box

The Design Reuse dialog box is launched from the Auto menu. The Select source schematic list is the tree view of the schematics in the Capture design. The sample design only contains one schematic, SCHEMATIC2 that also happens to be the reuse schematic. In the SCHEMATIC2 branch, two hierarchical blocks reference that schematic. Since the block that contains components U2 and R2 has been placed and routed, that entry is the source schematic. After REUSE2 is selected, all of the reuse schematics to which the source can be applied are listed in the Select target schematic(s) list. In this example, the only target available is REUSE1. If multiple targets were available in the design, all or some of them could be selected.

After selecting OK the target components are gathered and attached to the cursor for placement. Click to drop the placed and routed components on the board. If multiple targets were selected, after placing the first target the next set is attached to the cursor for placement. Press ESC at any time to cancel the placement of the additional targets.

Note: Undo is not supported in design reuse. It is not possible to undo any placement and routing information when using design reuse.

After placing REUSE1, the design looks should look like the following.



The design reuse portion is now complete.

External Design Reuse

External design reuse refers to reusing the placement and routing information of components from an external schematic and MAX file. External design reuse relies upon the fact that the referenced schematic has already been used for reuse design in another MAX file. A design with external design reuse could be as simple as a single schematic reference, like a power supply, or it could be more complex with multiple instances of the schematic in the design through hierarchical objects. The referenced schematic can be anywhere within the hierarchy of the external design.

Note: A reuse schematic includes all sub schematics within that schematic, unless a sub schematic is also a reuse schematic.

When a Capture design containing external design reuse is annotated, the part packaging information for the design reuse schematics is determined by the externally referenced schematics in the external design. External packaging information in the referenced schematic is preserved and used in the current design. The reference designators are changed to be appropriate for the current design, but the packaging stays the same.

Related topics

Internal Design Reuse

Partial Design Reuse

Creating the Capture design for external reuse

Design reuse begins with and is controlled by the organization of the Capture schematic. Items to be reused are grouped at the schematic level in Capture. All items contained within a reuse schematic are reused. The reuse schematic can exist anywhere within the design hierarchy.

Note: External design reuse relies upon the fact that the referenced schematic has already been used for reuse design in another MAX file.

Referencing a schematic that is in a Capture library:

It is possible to reference an external schematic that is in a Capture library. The design is annotated as though it is an internal design reuse schematic. This is because occurrence information is not preserved in a schematic when it is stored in a library. A referenced external schematic that is stored in a library will have default reference designators. It is also likely that the design will have duplicate reference designators and will need to be unconditionally re-annotated.

Copying and schematic from one Capture design to another:

Although you can copy a schematic from one Capture design to another for reuse, when the design is annotated, it is treated as an internal design reuse schematic. This happens because part packing information is not preserved when copying a schematic from one Capture design to another. After copying the completed schematic, the copy has default reference designators and it is likely that the design will have duplicate reference designators and need to be unconditionally re-annotated.

To create an external design reuse schematic:

1. Identify the schematic of the external design that you want to reuse.
2. Create the root schematic of your new design.
3. Place a hierarchical block in the root schematic page of the design, and reference the reuse schematic of the external design.
   - **a.** From the Place menu, choose Hierarchical Block
   - **b.** Enter a name for the hierarchical block in the Reference text box.

c. Choose Schematic View as the implementation type, in the Implementation Type list box.

d. Type the name of the schematic folder in the Implementation Name text box.

e. Since the reuse schematic is external, and not part of the current project, specify the path to the schematic folder in the Path and filename text box.

f. Click OK.

g. Use the cursor to draw the boundaries of the hierarchical block on the schematic page. Capture creates the new hierarchical block and automatically places the hierarchical pins according to the ports that exist in the attached schematic.

Related topics

Annotating the design for external reuse

Creating a netlist and starting a new board

Applying external design reuse to reuse targets

Annotating the design for external reuse

Once the Capture design has been drawn, wired, and footprints assigned, it needs to be annotated properly before creating a netlist.

The following are the annotation requirements for a Capture design that must be followed for design reuse to work properly.

- Duplicate references are not allowed in the design.
- Multi-part packages devices in design reuse schematics should not be packaged with multi-part packages outside of the design reuse schematic.
- Multi-part package devices should have consistent packaging across reuse schematics. This includes reference numbering and part packaging information.

The Layout Reuse tab in Capture's Annotate dialog box follows the above rules.

Note: For exceptions to the design reuse rules, see Partial Design Reuse.

To annotate the Capture schematic for external design reuse:

1. Open the project file in Capture.
2. Highlight the design file in the project manager.
3. From the Tools menu choose Annotate, and in the Annotate dialog box select the Layout Reuse tab.
4. In the Action group box, choose the appropriate annotate action. For more information about each action, see the Layout Reuse Tab (Capture Annotate dialog box) topic.
5. In the Physical Packaging group box enter the properties that must match for Capture to group the parts into a single package. The Value and Source Library properties are used as the default property string, but you can use any combination you like.
6. In the Select schematic(s) to mark for reuse: box, check all schematics that need to be marked for reuse. By selecting a schematic for reuse, it informs the annotation tool to treat this schematic (and all of its sub schematics) independently from the rest of the design. The reuse schematic is annotated following the required rules for Layout design reuse.
7. Click OK to annotate the schematic.
8. After the annotation is complete, check the Capture Session Log to see if there were any annotation errors. If there are errors after an incremental reference update, you may need to unconditionally annotate the schematic.

Related topics

Creating the Capture design for external reuse

Creating a netlist and starting a new board

Applying external design reuse to reuse targets

Creating a netlist and starting a new board

Note: Due to additions in the netlist format, you must use Capture 10.0 or newer to create the netlist. Older versions of Capture do not support Layout design reuse.

To create a netlist and start a new board:

1. Highlight the design file in the Capture project manager.
2. From the Tools menu choose Create Netlist.
3. Select the Layout tab in the Create Netlist dialog box and create the Layout netlist as usual.
4. In the Layout session window, choose New from the File menu to create a new board.
5. In the AutoECO dialog box choose the appropriate settings to create the new board file and click Apply ECO. For more information, see the AutoECO dialog box help topic.

Important

If you do gate swap or pin swap on a reuse block, you must do the same gate swap or pin swap on all instances of that reuse block, back annotate the changes to Capture, create the netlist for Layout in Capture, load the netlist into Layout, and then do design reuse.

Related topics

Creating the Capture design for external reuse

Annotating the design for external reuse

Applying external design reuse to reuse targets

To apply design reuse:

1. From the Auto menu choose Design Reuse. The Design Reuse dialog box appears.
2. Click Browse for source MAX file.
3. In the Open dialog box, navigate to the MAX file that was originally created from the source reuse schematic. The source MAX file should be already placed and routed.
4. Click Open. Notice that the schematic listed in the source list is updated to the new reuse schematic hierarchy.
5. In the Select source schematic box, navigate to the source schematic that you want to reuse. Click the check box by this schematic.

   Note: Layout determines design reuse based upon the organization of the source schematic. It is imperative to know the location of the source of reuse within that schematic.
6. In the Select target schematic(s) box, all of the available targets for design reuse are listed. Check all targets that you want to place and click OK.
7. The first instance of design reuse is now attached to your cursor. Click the left mouse button to place it. If you selected more than one target for reuse, the next target is automatically attached to the cursor. Press the ESC key at any time to cancel the placement of the additional targets.

   Note: Undo is not supported when using design reuse. You will not be able to undo any placement and routing applied with design reuse.

Related topics

Creating the Capture design for external reuse

Annotating the design for external reuse

Creating a netlist and starting a new board

External design reuse example

Creating the Capture design for external reuse

For this example, an external schematic in another Capture design is reused in the new design. The Capture design that contains the external schematic is similar to the design from the internal design reuse example. Gate assignments in the reuse schematic have been changed slightly to make it easier to see how gate packaging is preserved when referencing an external schematic. Also the root schematic contains Inverters instead of NAND gates.

The following is the external design file that contains the reuse schematic. The schematic that will be reused is SCHEMATIC2. Note that in this Capture design, the NAND

gates are packaged as B, C, and D, instead of the normally expected A, B, and C packaging.

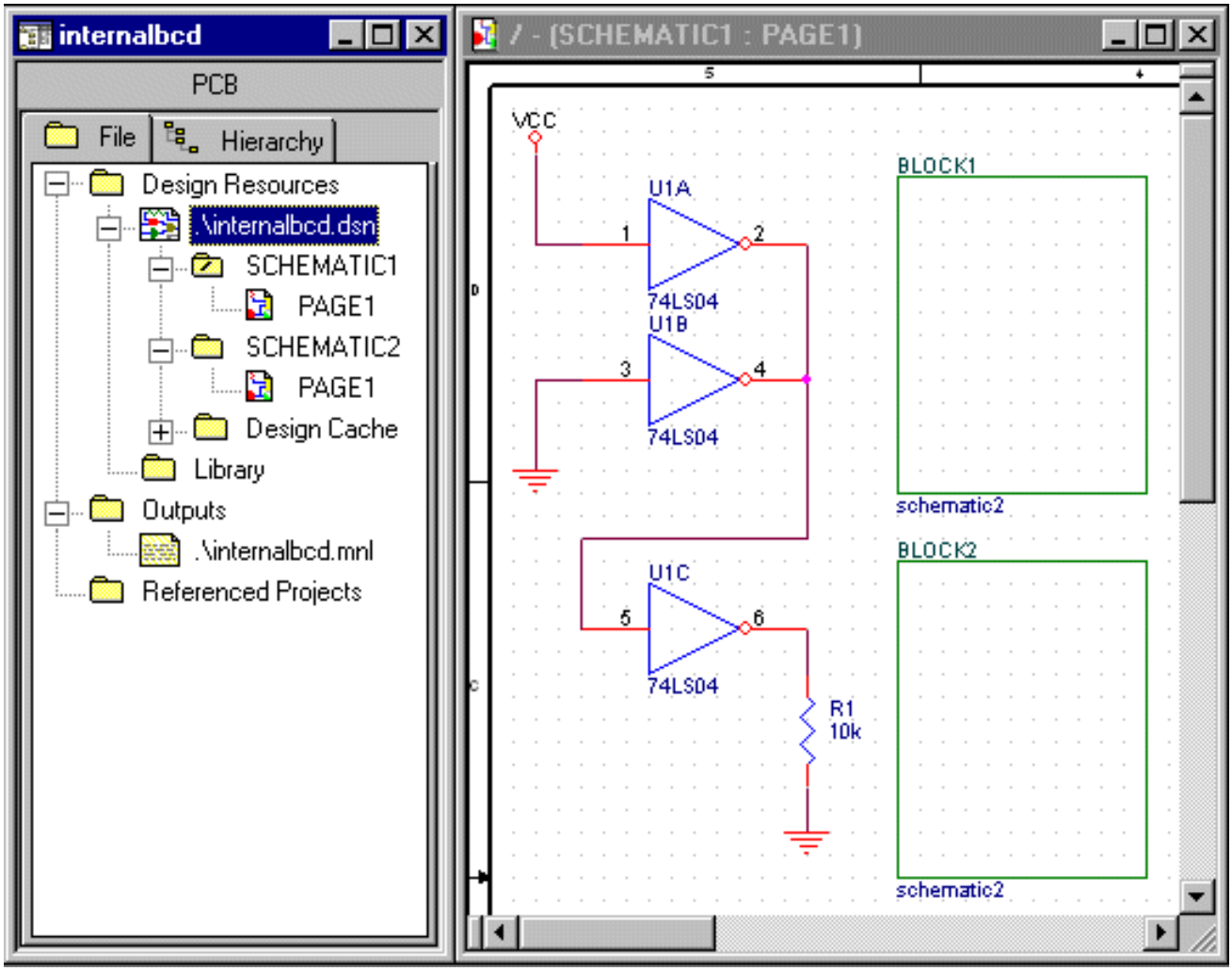


Figure 8-9 The external design file

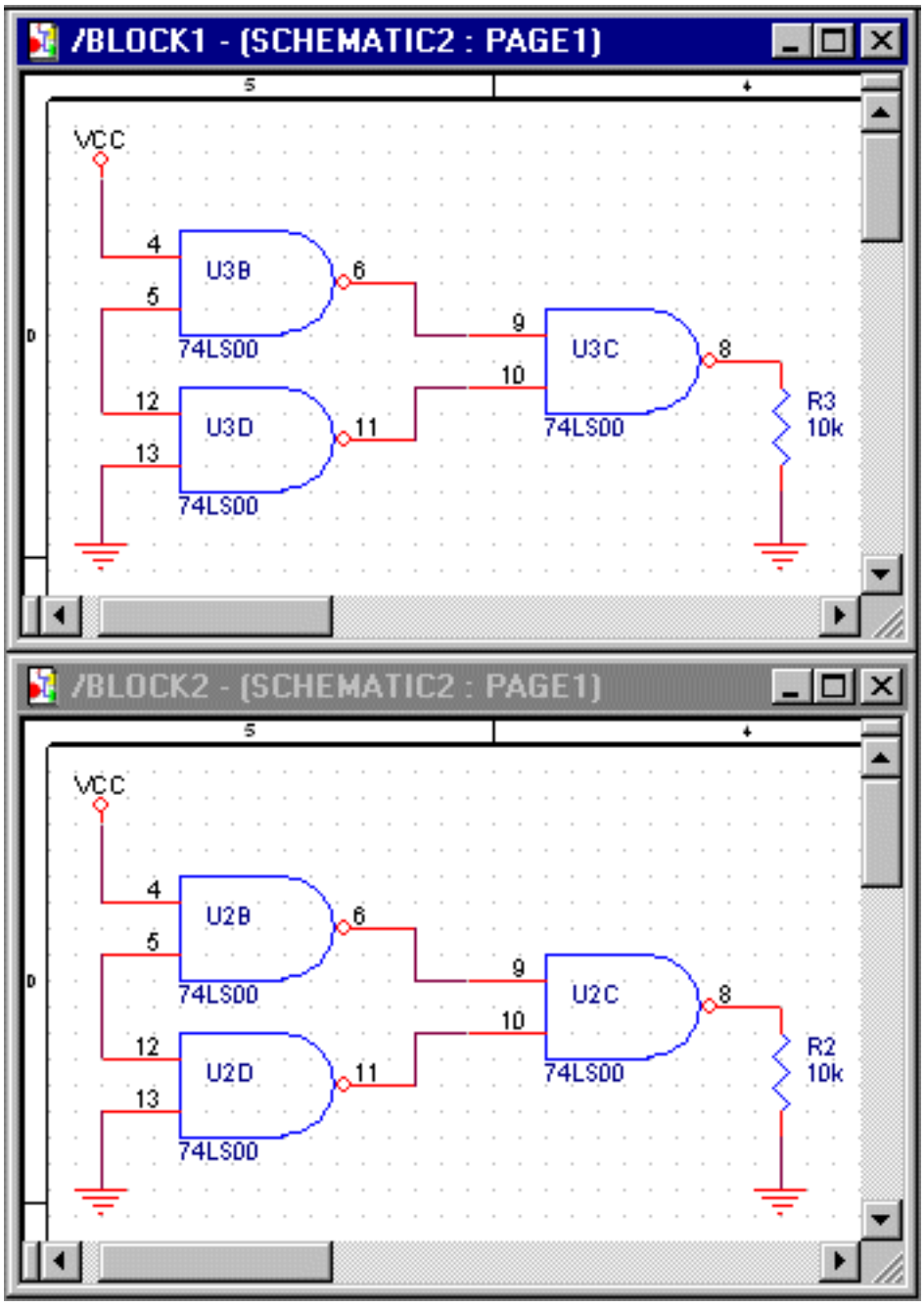


Figure 8-10 The schematic contained in the external design file. Note the irregular reference designators.

The following is a new design from which the above external schematic, SCHEMATIC2, is referenced. This design consists of a single root schematic, SCHEMATIC1 and two hierarchical blocks that reference the external reuse schematic, SCHEMATIC2.

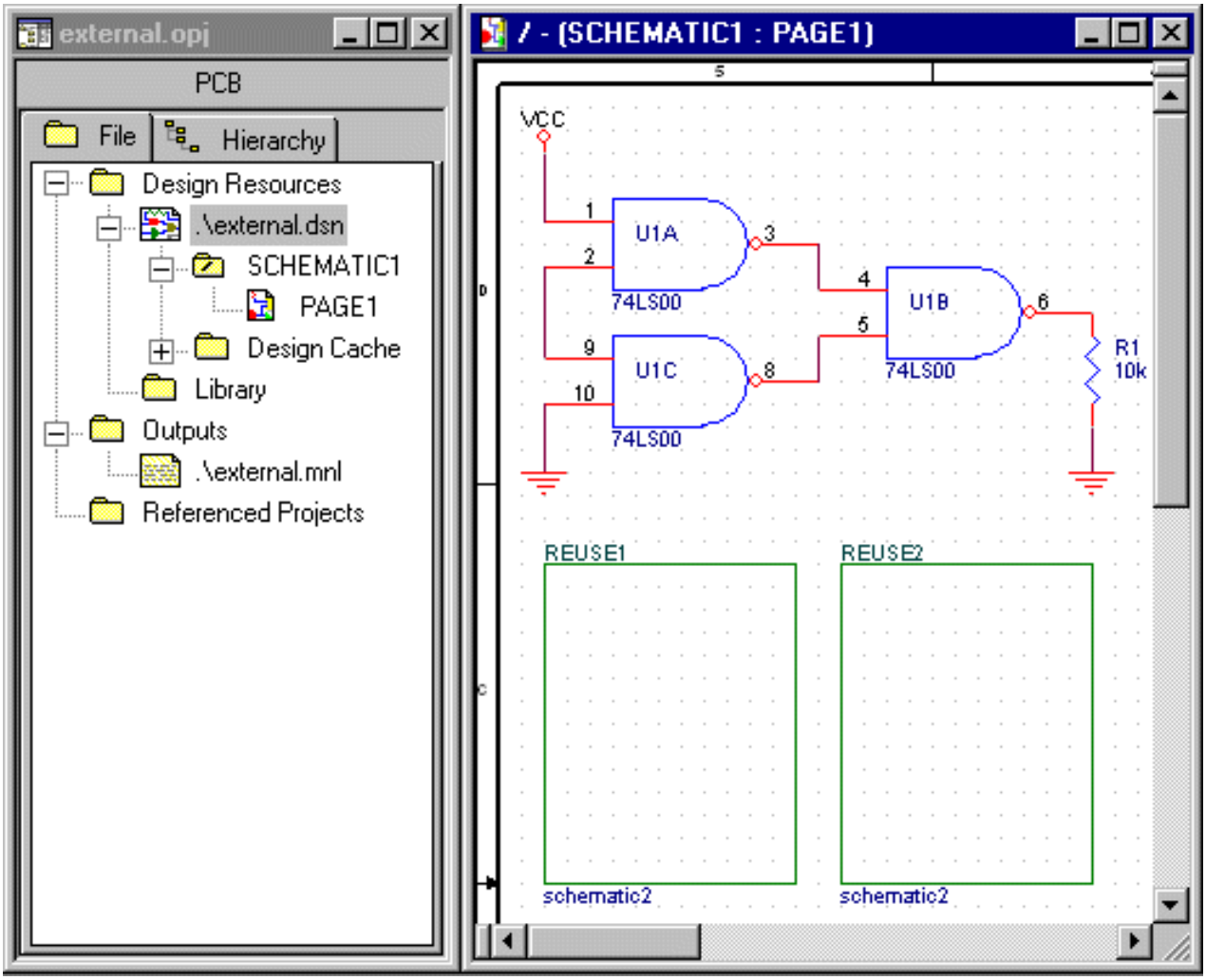


Figure 8-11 The hierarchical blocks in this design reference a schematic in an external design.

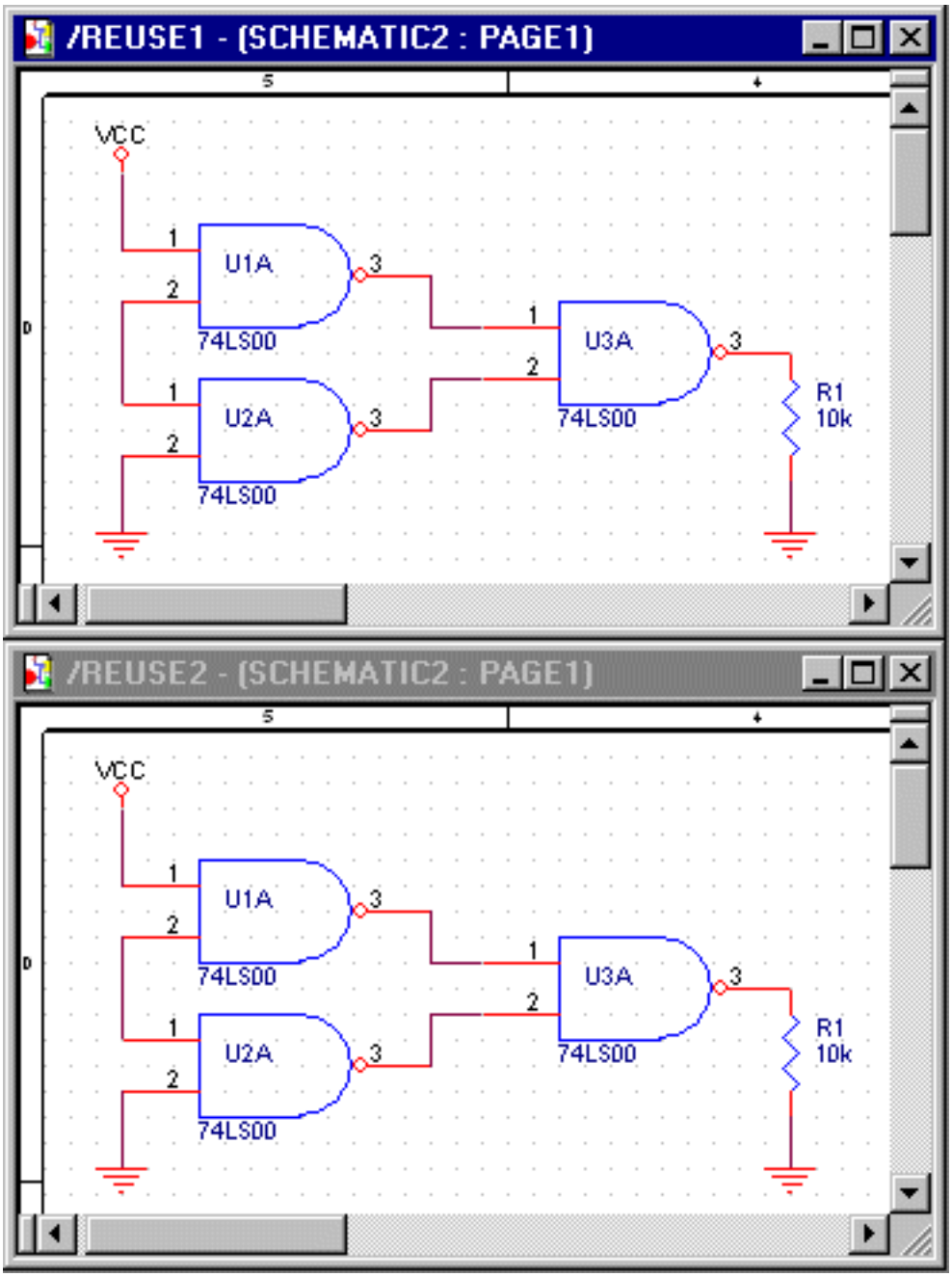


Figure 8-12 Schematics of the reuse blocks. Note the default values of the reference designators.

It is important to note that the reference designators in REUSE1 and REUSE2 are the default values for the design and are not the designators of the source. Because of duplicate reference designators, the design will need to be unconditionally annotated.

By checking the hierarchical block properties of REUSE1 it is possible to see how the block references the external Capture design and schematic. The Implementation value is the name of the reuse schematic, and the Implementation Path value is the name of the external Capture design.

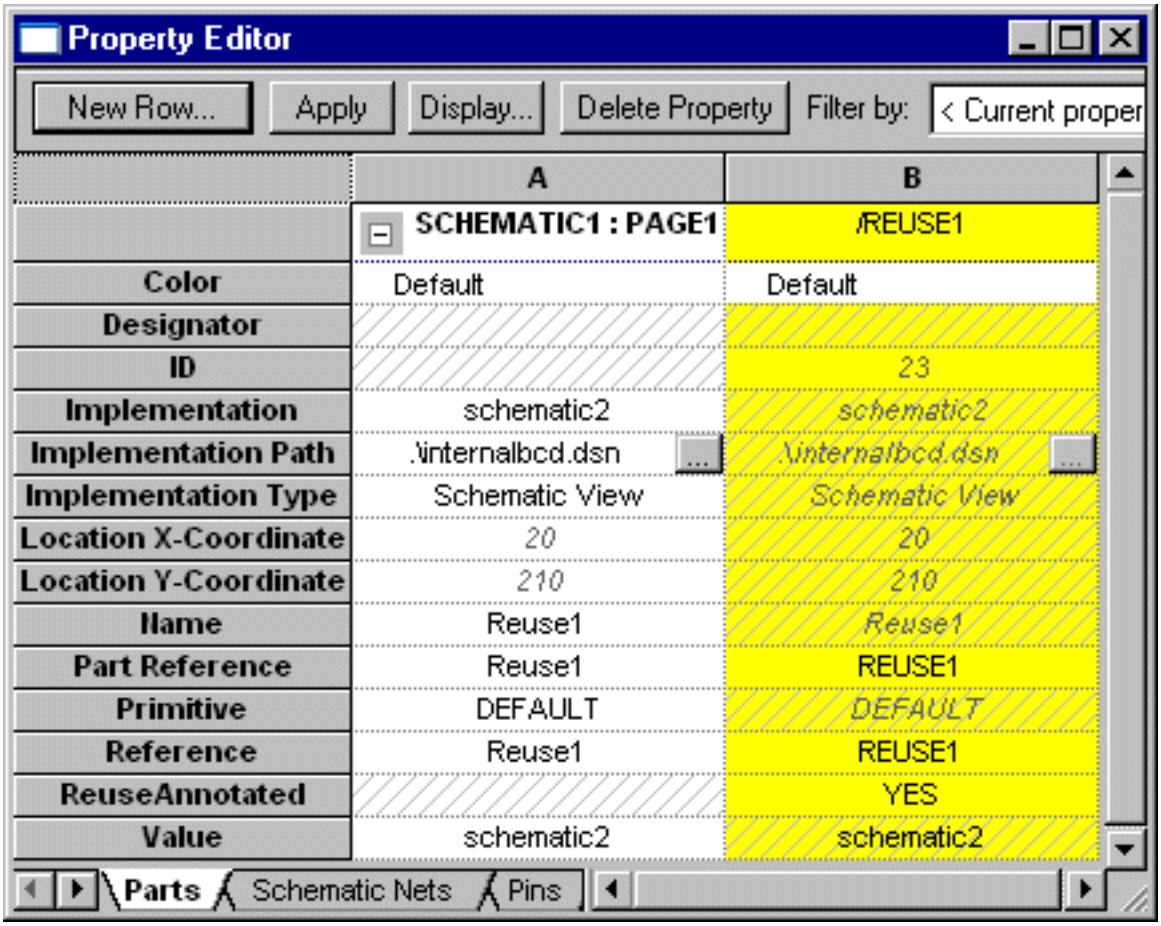


Figure 8-13 The Property Editor shows the implementation path to the referenced design and schematic.

Annotating the design for external reuse

Once the Capture design has been drawn, wired, and footprints assigned, it needs to be annotated properly. The Capture design does not currently meet the annotation design requirements for Layout design reuse, because it has duplicate reference designators. So, it is necessary to unconditionally annotate this design for Layout Reuse.

To annotate this design for reuse, select the Layout Reuse tab in the Annotate dialog box.

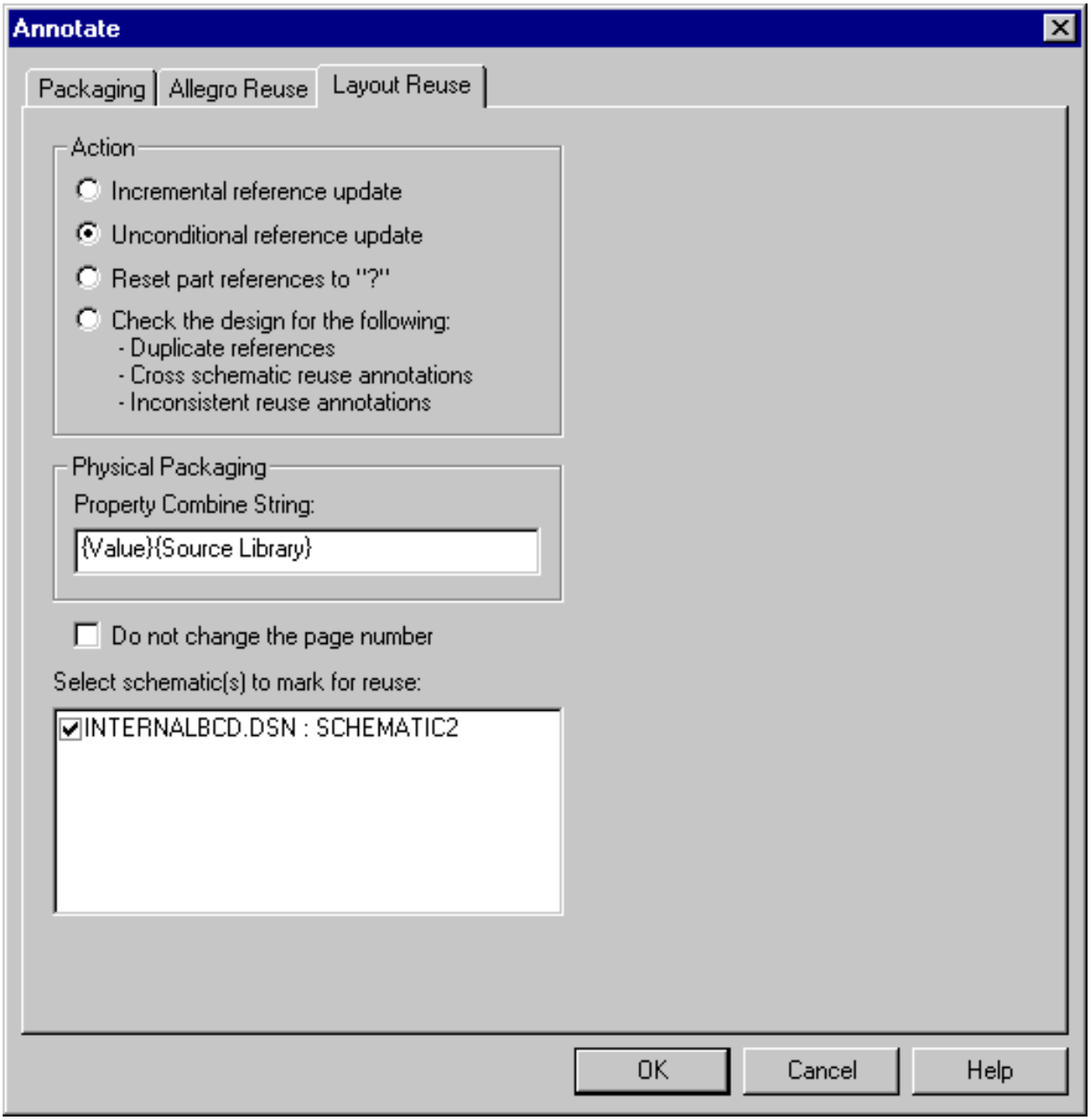


Figure 8-14 The Layout Reuse tab of the Annotate dialog box.

The above dialog shows the Layout Reuse tab configured to annotate this design properly. For this example, an Unconditional reference update has been chosen. Notice that the reuse schematic SCHEMATIC2 is referenced in the design INTERALBCD.DSN and has been selected in the Select schematic(s) to mark for reuse list. SCHEMATIC1 is not listed because it is the root schematic in the current design. By selecting SCHEMATIC2 for reuse, it informs the annotation tool to treat this schematic (and all of its sub schematics) independently from the rest of the design. The reuse schematic is annotated following the rules for Layout design reuse.

After the annotation is complete, the Capture Session Log indicates any errors. The following is the log file for the example design. The annotation was successful and no annotation errors were found.

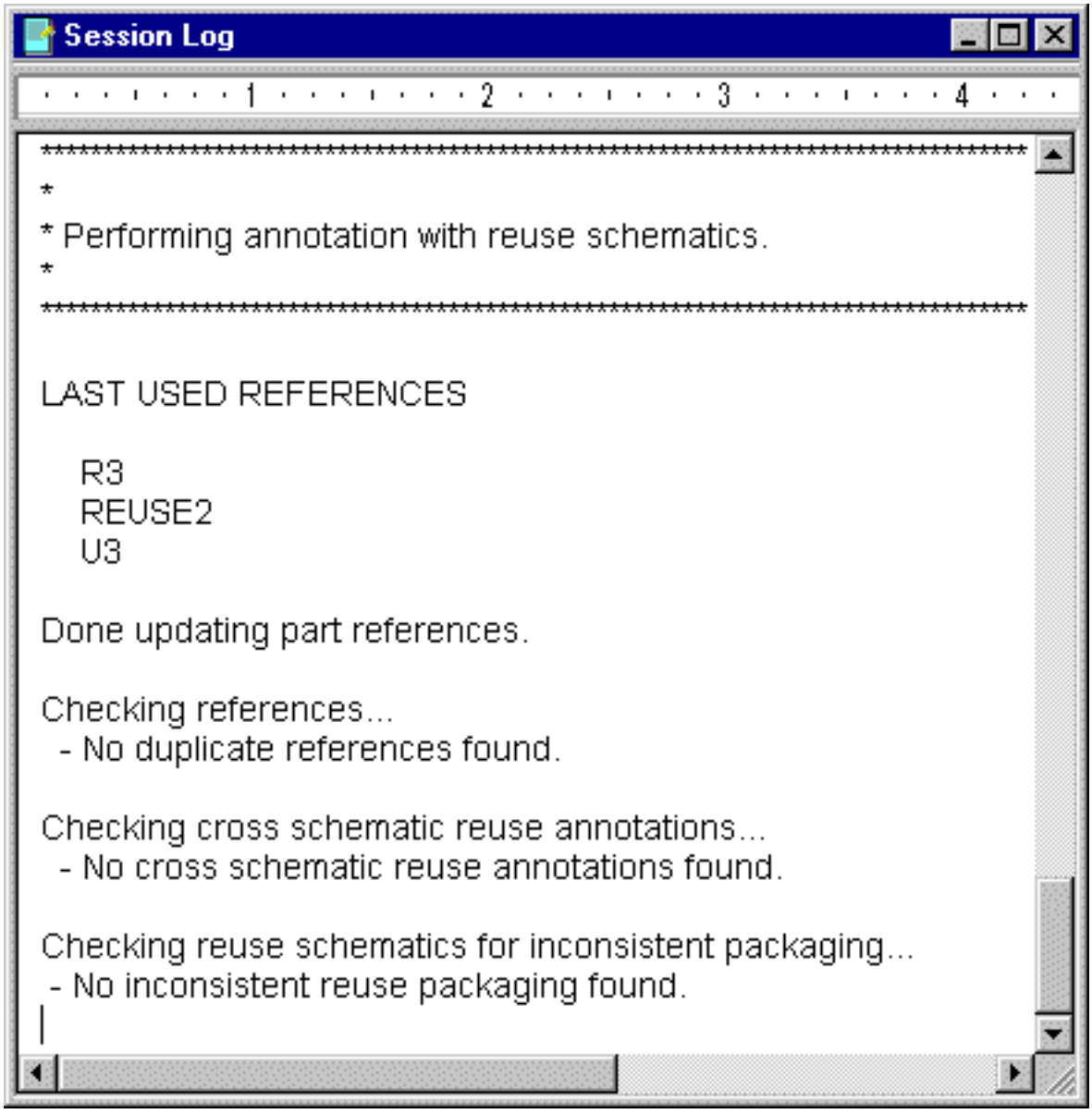


Figure 8-15 The Capture Session log indicates successful annotation.

The following shows the example design properly annotated for external design reuse.

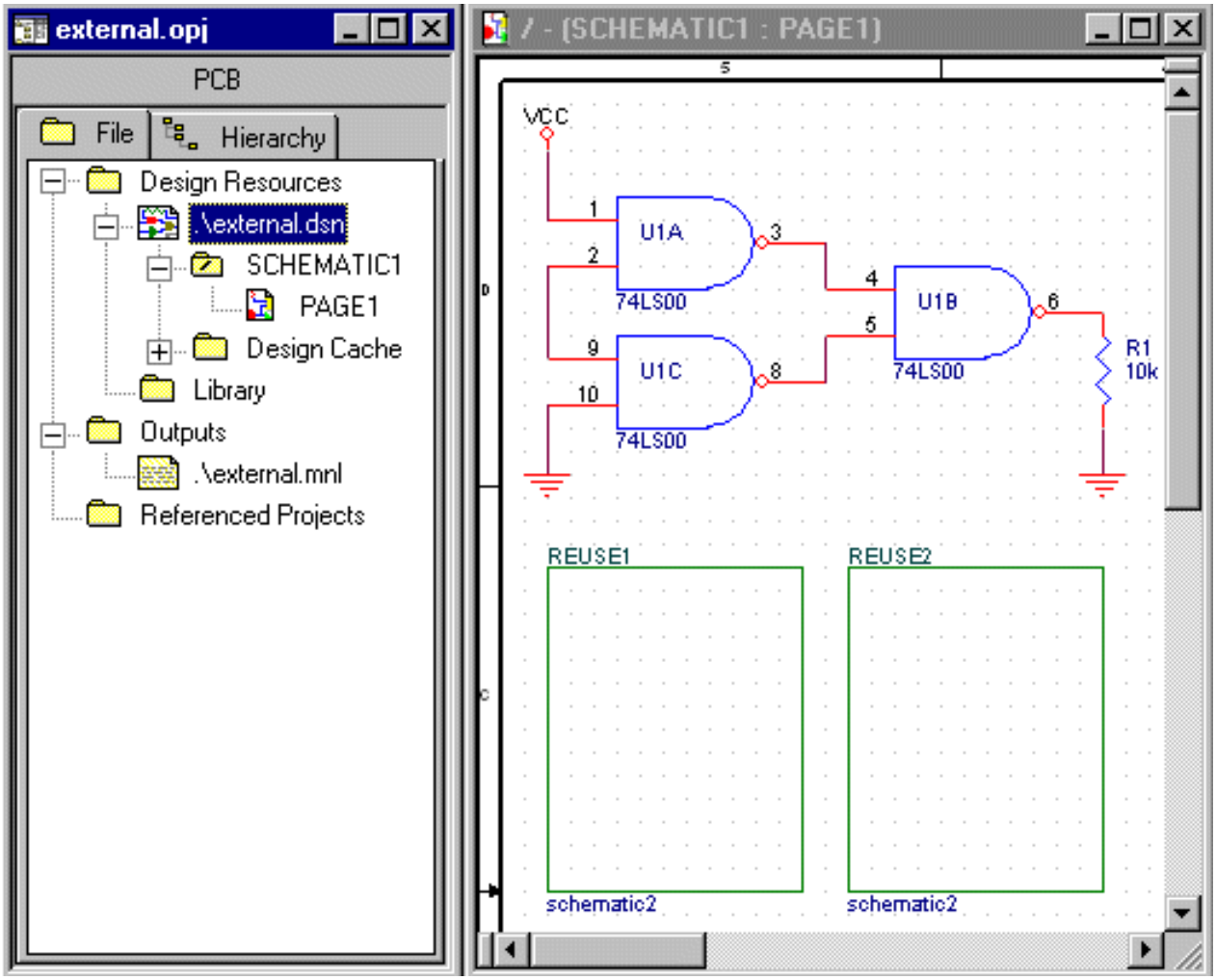


Figure 8-16 The correctly annotated design file.

Each reuse schematic (Reuse1 and Reuse2) has been packaged separately - from each other, and from the root schematic. Also note that in each reuse schematic, the corresponding parts have been assigned to the same

package, and match the reference designators in the reuse schematic.

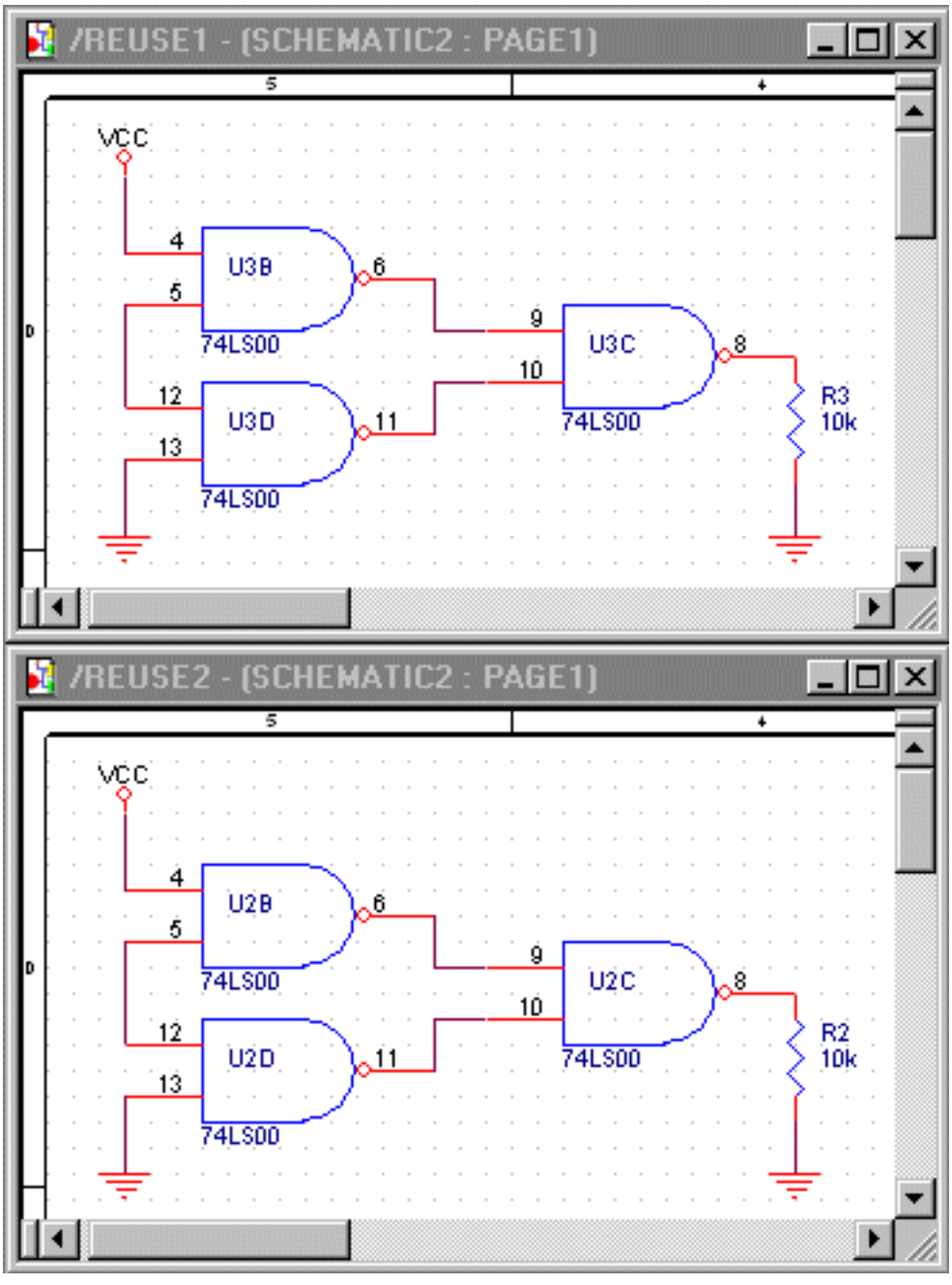


Figure 8-17 Correctly annotated external reuse schematics.

This design now meets the Layout reuse annotation requirements for Layout design reuse. It is now possible to create the Layout netlist.

Creating a netlist and starting a new board

To create a netlist for Layout design reuse, the Layout netlist is created as it would be for any other design.

Note: Capture 10.0 or newer must be used to create the netlist. Older versions of Capture do not support Layout design reuse.

If you do gate swap or pin swap on a reuse block, you must do the same gate swap or pin swap on all instances of that reuse block, back annotate the changes to Capture, create the netlist for Layout in Capture, load the netlist into Layout, and then do design reuse.

Applying external design reuse to reuse targets

The netlist is loaded into Layout and processed with the AutoECO utility. A board outline large enough to hold all of the parts is created, and then all of the components are block dragged into the outline. Since the components U1 and R1 are

not reused or the source of reuse, they are placed to the side out of the way.

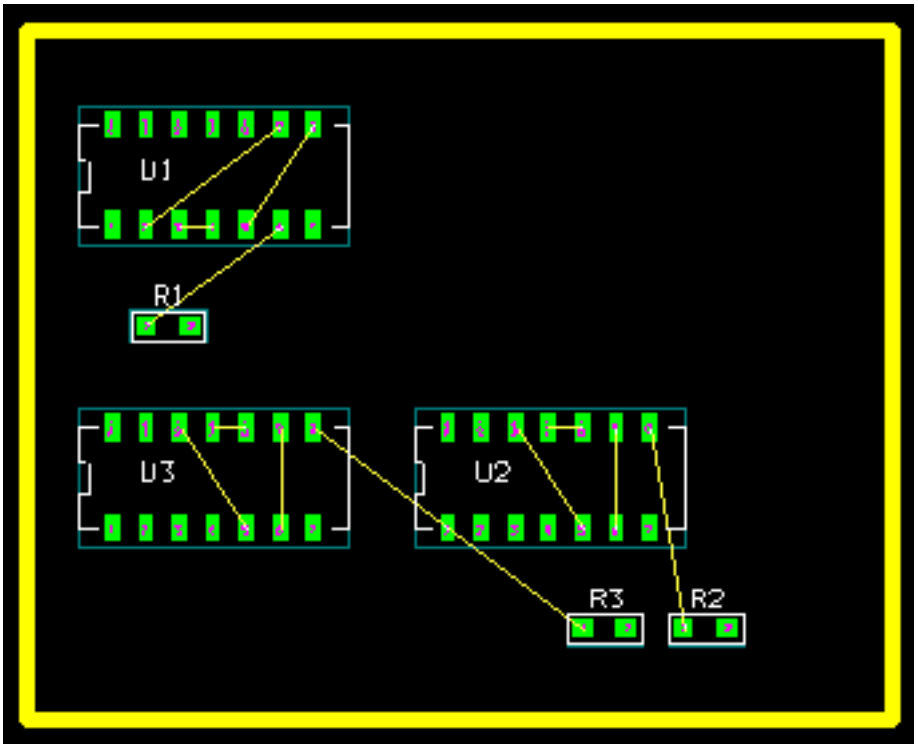


Figure 8-18 Layout MAX file with board outline and parts.

Note: The VCC and GND nets, and the DEFAULT ASYTOP layer have been disabled for clarity.

Since this is external design reuse, it is also necessary to have an external MAX file. The following MAX file was created from

the original external DSN file. The same reuse schematic was used in this MAX file as the current MAX file.

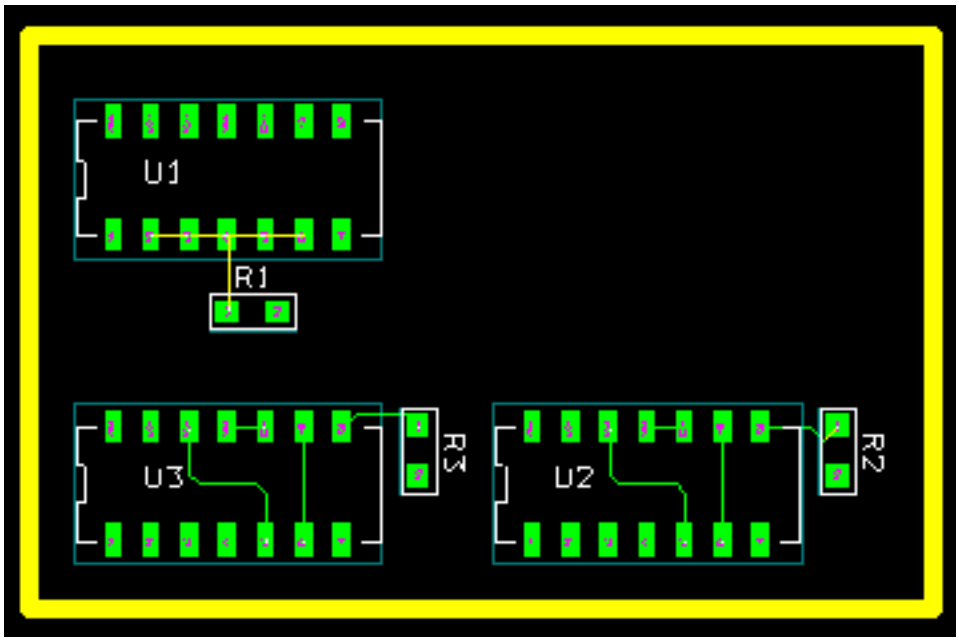
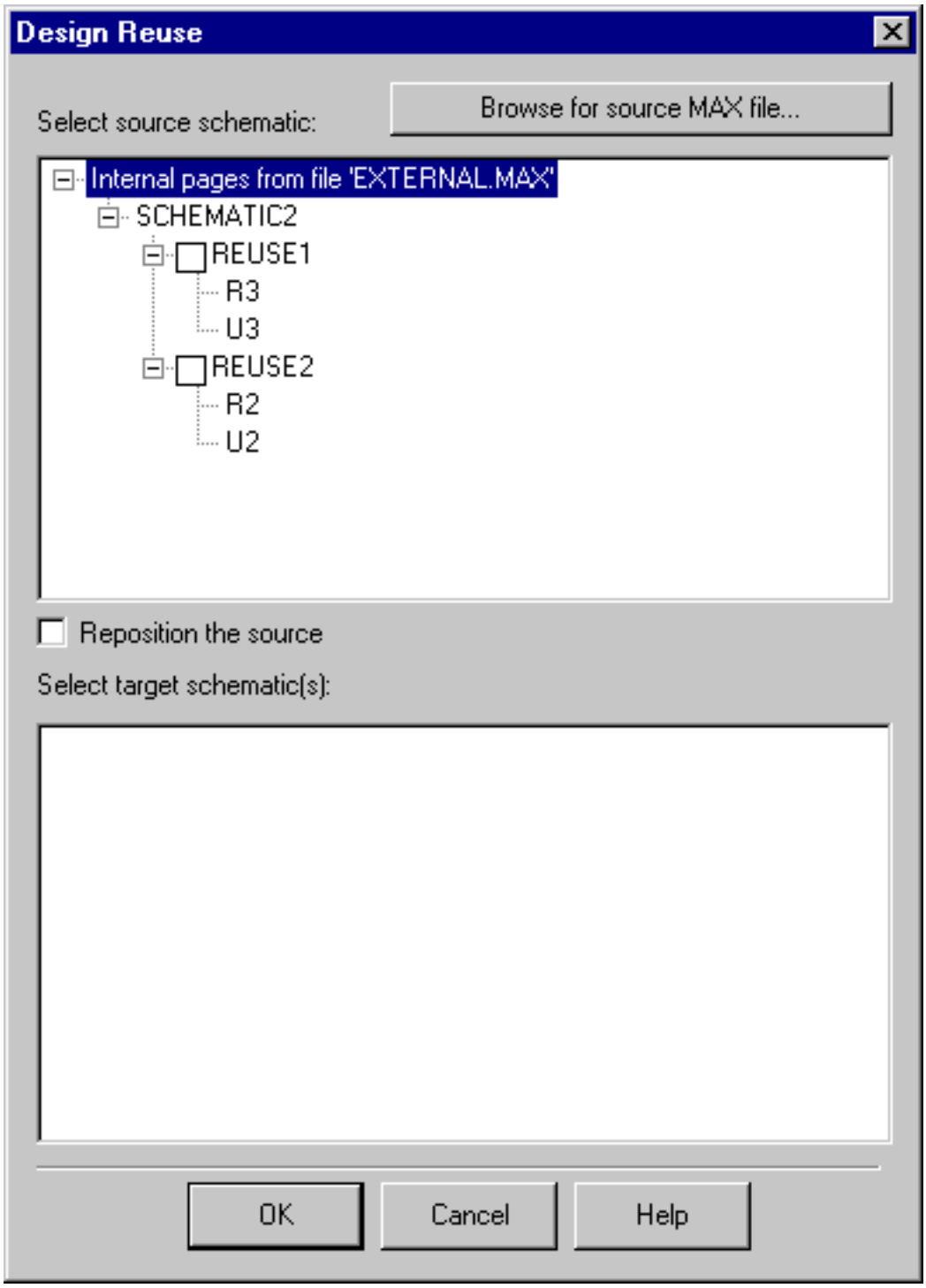


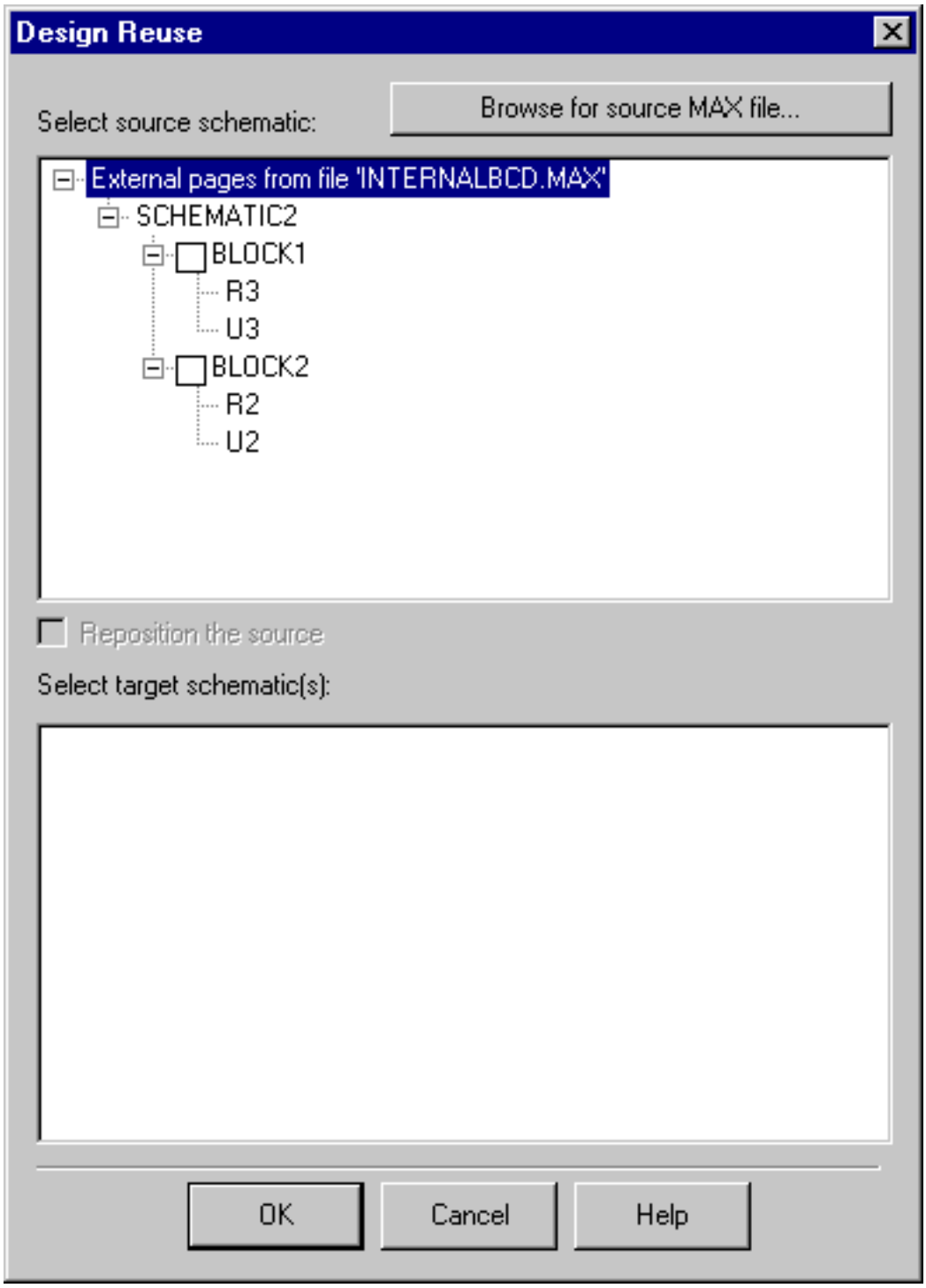
Figure 8-19 External design from which placement and routing is taken.

The Design Reuse dialog box is launched from the Auto menu. The Select source schematic list is the tree view of the schematics in the Capture design. By default, the Design Reuse dialog displays the hierarchy of the current design.

Since this is an external reuse design, it is necessary to use the hierarchy of a different MAX file.



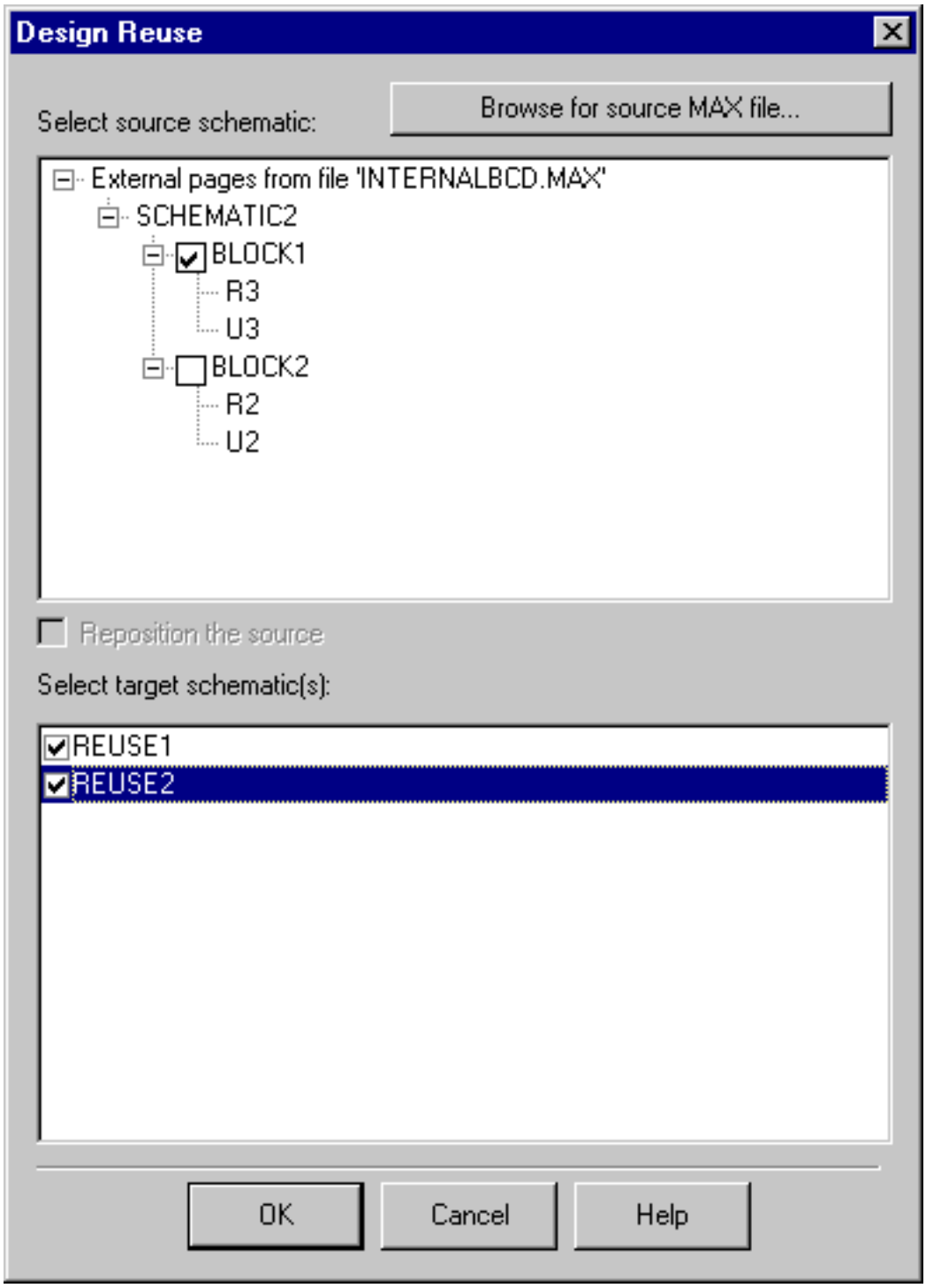
Using the Browse for source MAX file… button, the external MAX file is located and the associated hierarchy displayed.



The source schematic is now listed as "External pages from 'INTERNALBCD.MAX'". This is the schematic hierarchy from the external MAX file.

In the source list, there are two hierarchical blocks listed under SCHEMATIC2. Since each source hierarchical block is placed and routed identically, either block can be used as the source.

In this example BLOCK1 is used, but BLOCK2 will produce the same results.



After the source reuse schematic is chosen, the target list is updated to show all of the schematics that match the selected source. In this example, REUSE1 and REUSE2 match the source. In this example, design reuse will be applied to all targets, so both REUSE1 and REUSE2 are checked.

After clicking OK the target components are gathered and attached to the cursor for placement. Click to drop the placed and routed components on the board. Since multiple targets were selected, after the first target is placed, the next group of components is attached to the cursor for placement. Press ESC at any time to cancel the placement of the additional targets.

Note: Undo is not supported in design reuse. It is not possible to undo any placement and routing information when using design reuse.

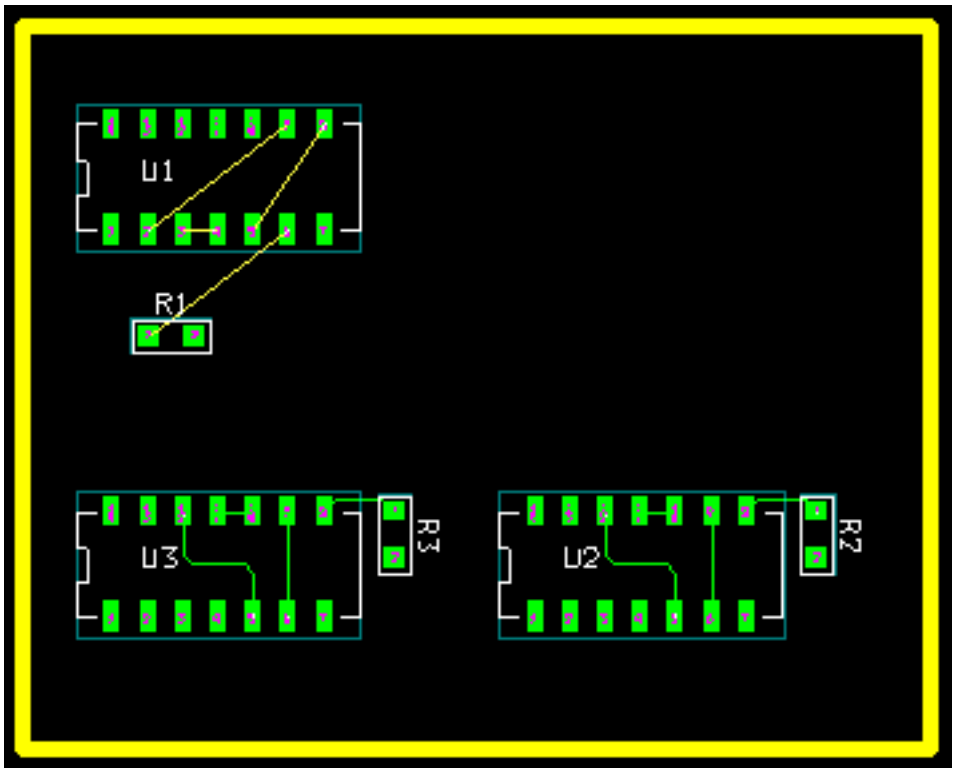


Figure 8-20 The Layout MAX file after design reuse has been applied.

The design reuse portion is now complete.

Partial Design Reuse

It is possible to break the design reuse rules. Doing so creates a design that only contains partial reuse. With partial design reuse, you must follow these practices for design reuse to be effective:

1. Annotate the design with the Layout Design Reuse tab of the Capture Annotate dialog box.
2. Hand annotate all exceptions to design reuse in Capture.
3. Manually place and route the components that are design reuse exceptions in Layout.

For example, if you want to have a part that shares gates between a reuse schematic and a non reuse schematic this creates partial design reuse. The shared part is not included in design reuse. The part will need to be hand annotated in Capture, and manually placed and routed in Layout.

If you do gate swap or pin swap on a reuse block, you must do the same gate swap or pin swap on all instances of that reuse block, back annotate the changes to Capture, create the netlist for Layout in Capture, load the netlist into Layout, and then do design reuse.

To implement partial design reuse:

1. Create the reuse schematic.
2. Annotate the schematic with the Layout Reuse tab. For more information, see the Layout Reuse tab (Capture Annotate dialog box) topic.
3. Hand annotate any exceptions to the design reuse rules.
4. From the Layout Reuse tab, choose the Check the design for the following Action and click OK.

5 In the Capture Session Log, examine the errors and warnings reported. The only warnings reported should be your hand annotations.

6 Create the Layout netlist.

7 Create the Layout MAX file from the netlist.

8 From the Auto menu choose design reuse.

9 Place all instances of design reuse.

Note: Partial design reuse components are not included.

10 Manually route each partial design reuse component.

Related topics

Internal Design Reuse

External Design Reuse

Placing and editing components

9

After you set up your board, you can begin placing components. Whether you are placing components manually, or using the autoplacement feature in Layout Plus, you can place components individually or in groups, and can take advantage of a variety of powerful placement commands. The steps involved in the component placement process are listed below.

- Optimize the board for component placement
- Load a placement strategy file
- Place the components on the board
- Optimize placement using various placement commands

Preparing the board for component placement

Before you begin placing components manually, it is important to set up the board properly. Use the list below as a preplacement checklist.

- Check the board, place, and insertion outlines
- Check the place grid
- Check mirror layer or library layer settings
- Weight and color-code nets

- Check gate and pin data
- Check preplaced components and secure them on the board using the Lock or Fix commands
- Create component height keepins and keepouts, or group keepins and keepouts

Checking the board, place, and insertion outlines

The board outline is used by Layout to determine the overall board placement boundary, and it must be present on the global layer of the board. It can be defined as part of the board template, or you can create it when you set up the board.

A place outline defines the extent of the area that is reserved for a component's placement. Each footprint must have one. Layout uses place outlines to determine whether component spacing violations occur during placement.

Cadence recommends that you use only one place outline for a footprint. The height and shape of the place outline must closely represent the placement area required by a component.

A place outline can be assigned a height and a layer. You can use one place outline on each layer. The place outline on each layer can have different height and shape, to more closely represent the placement area required by a component.

Note: If you select the Show 3D Effects option in the User Preferences dialog box (accessed by choosing User Preferences from the Options menu), and have assigned a height for a place outline, Layout displays a three-dimensional image representing the component's height, and indicates the height on the image.

An insertion outline is optional, and is used by Layout to provide clearance for auto-insertion machines.

Note: An insertion outline can overlap another insertion outline, but a place outline cannot overlap another place outline.

To check board, place, and insertion outlines

1. Choose the spreadsheet toolbar button, then choose Obstacles. The Obstacles spreadsheet appears.
2. Review the Obstacle Type column in the spreadsheet to check that the board, place, and insertion outlines have the correct width and height, and that they are on the correct layer (for example, the board outline must be on the global layer).
3. Close the Obstacles spreadsheet so that you can view the board outline in the design window. If there are "cutouts" in the board outline where no components should be placed, you need to create zero-height keepouts inside the cutouts, to ensure that no components are placed in these areas.

Note: For information on creating height keepouts, see *Creating height or group keepins and keepouts*. For information on creating board outlines, see *Creating a board outline*.

Checking the place grid

The place grid affects the spacing used for component placement. Before placing components, check the setting for the place grid in the System Settings dialog box.

The default placement grid is 100 mils, with which you can use routing grids of 25 mils, 20 mils, 121/2 mils, 10 mils, 81/3 mils, 61/4 mils, or 5 mils (because 100 mils is a multiple of these values).

Note: If you use a 50 mils or 25 mils placement grid, you can use routing grids of 25 mils, $12^1/_2$ mils, 10 mils, $8^1/_3$ mils, or $6^1/_4$ mils.

The standard metric placement grids are 2 mm, 1 mm, and 0.5 mm.

To check the place grid setting

1. From the Options menu, choose System Settings. The System Settings dialog box appears.
2. Check the value in the Place grid text box, change it if necessary, then choose the OK button.

Checking mirror layers and library layers

You can check which layers are set up to have their obstacles, padstacks, and text mirrored to another layer during component placement, and change the settings, if necessary. For example, all of the TOP layer components can be automatically mirrored to the BOTTOM layer, and vice versa.

Typically, all inner layers of a design (INNER1, INNER2, and so on) correspond to the INNER library name, and all plane layers of a design (POWER, GROUND) correspond to the PLANE library name. All other layers typically have a one-to-one correspondence; for example, the BOTTOM layer in the design corresponds to the BOTTOM library name.

To check the mirror layer and library layer settings

1. Choose the spreadsheet toolbar button, then choose Layers. The Layers spreadsheet appears.
2. Check the settings in the Mirror Layer column against the settings in the Layer Name column, to ensure that the layers are set to mirror to their opposite layers.
3. Double-click on each layer to bring up the Edit Layer dialog box, check that the Layer LibName is set appropriately, then press ESC to close the dialog box.

Weighting and color-coding nets

Layout places a higher priority on keeping higher-weighted nets and their components together during placement. In Layout, nets are weighted on a linear scale from 0 to 100.

To weight and highlight nets

1. Choose the spreadsheet toolbar button, then choose Nets. The Nets spreadsheet appears.
2. Double-click in the Net Name cell that corresponds to a net whose weight you want to change, or that you want to highlight. The Edit Net dialog box appears.
3. To change the weight for a net, type in a new weight in the Weight text box, then choose the OK button.
   or
   Use the scroll bar at the left of the text box to change the number, then choose the OK button.
   The new number shows in the Weight column of the spreadsheet.
4. To highlight a net, select the Highlight option in the Edit Net dialog box, then choose the OK button. The net shows in the highlight color.

Note: To assign a color to a net other than the highlight color, click in the Color cell in the Nets spreadsheet, choose Change Color from the pop-up menu, then select a color from the color palette that appears.

Note: For information on setting net properties, see *Chapter 5, "Setting up the board."*

To color-code a net

1. In the Nets spreadsheet, select the net(s) to which you want to assign a color.
2. From the pop-up menu, choose Change Color, then select a color from the color palette that appears.

Checking gate and pin information

A package is the electronic gate and pin information associated with a component (as opposed to a footprint, which is the information regarding the physical characteristics of a component). The information in the Packages spreadsheet is used to determine whether you can swap gates between

identical components or only within a component, and how the gates are arranged within a part.

To check gate and pin information

1. Choose the spreadsheet toolbar button, then choose Packages. The Packages spreadsheet appears.
2. Verify that the following information in the spreadsheet is correct, then close the spreadsheet.

Package Name

A text string that designates the name of the electrical package.

Gate Name

Usually an alpha character that designates which gate each pin belongs to. Each gate in a package must have a unique gate name, and all of the pins in the same gate must share the same gate name.

Pin Name

Identifies each pin in terms of its electrical characteristics (INA, INB, and so on) so that Layout can swap gates correctly. Each pin within a gate must have a unique identifier. For swappable gates, corresponding pins must have identical pin names.

Gate Group

An integer used to determine which gates can be swapped. Any gates that are assigned to the same Gate Group are swappable. Gate Group 0 is a special case that represents a non-swappable gate.

Pin Group

An integer used to determine which pins can be swapped. Any pins that are assigned to the same Pin Group are swappable. Pin Group 0 is a special case that represents a non-swappable pin.

Pin Type

Usually set to None for standard TTL-type pins, which indicates that the pin is not part of an ECL net, and is not a source, a terminator, or a load. You can assign a Pin Type of None, Source, Terminator, or Load.

Securing preplaced components on the board

If your design has components or footprints that were placed at the schematic level or as part of the template, you should ensure that they were placed properly before you begin placing additional components. Preplaced components may include connectors, mounting holes, memory arrays, predefined circuits, alignment targets, and components that must be placed in specific locations due to mechanical or temperature restrictions.

After you are satisfied that the preplaced components are properly placed, you must affix them to the board using the Fix or Lock commands. Otherwise, they may be moved inadvertently when you are placing other components.

The Lock command is temporary; you can easily override the command. However, the Fix command must be disabled in the Edit Component dialog box. The Fix command is intended for parts like connectors and mounting holes that need to be placed permanently in specific locations.

To secure components on the board

1. Choose the component toolbar button.
2. To select all of the preplaced components, hold the left mouse button down while you drag the mouse, drawing a

rectangle around the components. Release the left mouse button. Each selected component is highlighted.

3 To temporarily lock components at a location, choose Lock from the pop-up menu.
or
To permanently fix components at a location, choose Fix from the pop-up menu.

To override the Lock command

1 Select a few locked components. A dialog box asking "One or more components locked. Override?" appears.

2 Choose the OK button. The components are unlocked.

To override the Fix command

1 Choose the spreadsheet toolbar button, then choose Components. The Components spreadsheet appears.

2 Double-click on the row for the component that you want to move. The Edit Component dialog box appears.

3 In the Component flags group box, deselect the Fixed option, then choose the OK button.

Creating height or group keepins and keepouts

You can restrict component placement based on physical constraints using the Comp height keepin or Comp height keepout obstacle types. A height keepin contains all components at or above a specified height, while a height keepout excludes all components at or above a specified height.

You can also restrict placement based on group number (assigned in the schematic) using the Comp group keepin or Comp group keepout obstacle types. A group keepin contains all the components in a specified group, while a group keepout excludes all the components in a specified group.

To create keepins and keepouts

1. Choose the obstacle toolbar button.
2. From the pop-up menu, choose New.
3. Draw a rectangle that defines the desired keepin or keepout area.
4. Double-click on the rectangle. The Edit Obstacle dialog box appears.
5. In the Obstacle Type drop-down list, select Comp height keepin or Comp height keepout.
6. In the Height text box, enter a number corresponding to the height of the components you want to include or exclude, then choose the OK button.
   or
   In the Obstacle Type drop-down list, select Comp group keepin or Comp group keepout. In the Group text box, enter a number corresponding to the group number of the components you want to include or exclude, then choose the OK button.
7. From the pop-up menu, choose Finish. If you created a component height restriction, the rectangle displays the height number and the words “Comp keepin” or “Comp keepout.”
   or
   From the pop-up menu, choose Finish. If you created a component group restriction, the rectangle displays the group number and the words “Group *number* keepin” or “Group *number* keepout.”

Note: If your keepins and keepouts don’t display any identifying text (as described in step 7), you may have to enable the Show 3D Effects option. To do so, choose User Preferences from the Options menu. In the User Preferences dialog box, select the Show 3D Effects option, then choose the OK button.

Loading a placement strategy file

Strategy files set up your screen display by highlighting appropriate elements such as place outlines, electrical

connections, and reference designators, and making irrelevant elements (such as plane layers) invisible. OrCAD recommends loading the strategy file PLSTD.SF before performing manual placement.

To load a placement strategy file

1. From the File menu, choose Load. The Load File dialog box appears.
2. If necessary, change Files of type to Strategy.
3. Select PLSTD.SF from the list and choose the Open button.

Grouping components for placement

You can assign components as a group while you are working in Capture before you create a netlist, or while you are working in Layout by using the Components spreadsheet. The preferable approach is to assign groups while you are working in Capture.

In Capture, for each component in the group, you need to generate a user-defined part property with property name COMPGROUP and a common integer as the property value. See the Capture documentation for detailed instructions.

If you are working with Layout Plus, consider that clustering components yields quicker placement than grouping does.

To group components in Layout

1. In the Components spreadsheet, select the components that you want to group.
2. From the pop up menu, choose the Properties command to display the Edit Component dialog box.
3. Assign an integer as a group number and choose the OK button.

Related topics

Building component clusters

Edit Component dialog box

Component command (Place)

Matrix command (Place)

Disabling the power and ground nets

If the power and ground nets are not critical to placement, disable routing for all nets attached to plane layers. This significantly improves system performance during placement, because these (typically) large nets often have no bearing on placement.

To disable routing for nets attached to plane layers

1. Choose the spreadsheet toolbar button, then choose Nets. The Nets spreadsheet appears.
2. Using the CTRL key, select the nets that are attached to plane layers (usually, GND and VCC).
3. From the pop-up menu, choose Enable<->Disable. In the Nets spreadsheet, the Routing Enabled column for the nets changes to No.

Cross-placing components from Capture to Layout

The cross-placement feature lets you select a component in OrCAD Capture and place it in Layout. This lets you quickly place components on the board.

When you select a component in Capture and move the cursor over the Layout window, the component is attached to the cursor in a place (or move) mode.

To cross-place components from Capture to Layout

1. Open the schematic in Capture and the board in Layout.
2. Ensure that intertool communication (ITC) is enabled in Capture.
   a. From the Options menu in Capture, choose Preferences.

      The Preferences dialog box appears.

   b. Click the Miscellaneous tab and ensure that the Enable Intertool Communication check box is selected.

   c. Click OK.
3. From the Options menu in Layout, choose User Preferences.

   The User Preferences dialog box appears.
4. Select the Place component from Capture check box and click OK.

   Note: Layout remembers this setting only for the current session. The next time you start Layout, you have to select this check box again if you want to cross-place components from Capture to Layout.

Tip

Running Layout in half screen mode makes it easier to cross-place components from Capture to Layout. From the Window menu in Layout, choose Half Screen to run Layout in half screen mode.

5. In Layout, zoom in to the location in the board where you want to place the component.
6. Select the component in Capture and move the cursor over the Layout window

   The component is attached to the cursor in Layout. Place or move the component as required.

   Note the following:

- If the component you selected in Capture is locked in Layout, Layout allows you to override the lock and cross-place the component.
- If the component you selected in Capture is fixed in Layout, Layout will not allow you to cross-place the component and displays a message that the component is fixed.

For more information on locking or fixing components, see Securing preplaced components on the board on page 329.

Placing components manually

There are several commands available in Layout to assist you in manually placing components on a board. You can place components one at a time or in groups.

Note: Before you begin placing components, save your board file.

Use the Queue For Placement command to make a component or group of components available for placement based on a set of criteria (reference designator, footprint name, or first letters with wildcards), then place the components individually using the Select Next command.

Placing components individually

To place components individually

1. Choose the component toolbar button.
2. From the pop-up menu, choose Queue For Placement. The Component Selection Criteria dialog box appears.

   Note: The Queue For Placement command and the Select Any command display the same Component Selection Criteria dialog box, but the commands work differently. The Queue For Placement command makes certain components available for placement in conjunction with using the Select Next command. The

Select Any command, on the other hand, actually selects specified components or groups for placement and attaches them to your cursor.

3 Enter the reference designator (or other criteria) of the component that you want to place in the appropriate text box, then choose the OK button. (Choose the dialog box's Help button for information on the options in the dialog box.)

Note: You can specify more than one component using wildcards: use an asterisk (*) as a substitute for multiple characters and a question mark (?) as a substitute for a single character. For example, if you enter U*, you will select all components with reference designators beginning with the letter U.

4 From the Edit menu, choose Select Next. The component snaps to the cursor. If you selected a group (such as all components beginning with the letter *U*), then the component with the greatest number of connections that meets the specification snaps to the cursor.

5 Drag the component to the desired location and click the left mouse button to place it.

Selecting the next components for placement

Use the Place command on the pop-up menu to display a dialog box that lists the components yet to be placed. If you made components available for placement according to certain criteria (using the Component Selection Criteria dialog box), Layout displays only the components that remain to be placed that meet those criteria. From this list, you can select the next component that you want to place.

The default selection that appears in the dialog box is the one that Layout would automatically choose if you had used the Select Next command. You can accept the default, or enter a new choice.

To select the next component for placement using Select Next

1. Choose the component toolbar button.
2. From the pop-up menu, choose Place. The Select Next dialog box appears.
3. Select a component for placement, then choose the OK button.

Building component clusters

A cluster is represented on screen (with Reconnect Mode enabled) as a circle that has an area equal to the area of the components within the cluster, and is automatically given the name (ID) of the largest component within the cluster.

After components have been assigned to clusters, you can choose Select Next (N key) from the pop-up menu to select the most heavily interconnected components from the clusters for placement on the board.

You can also select a particular cluster and press N if you want components from that cluster for placement first.

To build a component cluster

1. Grouping components for placement that need to be in the cluster.
2. In the design window, select the Reconnect Mode and the Component Tool buttons on the tool bar.
3. From the Edit menu, choose the Select Any command to display the Component Selection Criteria dialog box.
4. In the Group Number text box, enter the number that you assigned to the group, then choose the OK button to dismiss the dialog box.
5. From the pop-up menu, choose Make. Wait for the components to jump to the pointer.
6. Move the pointer to position the cluster and click the left mouse button to place the cluster.

7 From the pop-up menu, choose the Break command to release the individual components from the cluster. Now you can select each component in turn and place it at the optimal location.

Shortcuts

Keyboard: K

Related topics

Edit Component dialog box

Component command (Place)

Matrix command (Place)

Placing component groups

You can assign functionally related components to groups at the schematic level. When you specify the group number (as assigned in the schematic) in the Component Selection Criteria dialog box, the components assigned to the group snap to the cursor for placement.

To place a component group

1 Choose the component toolbar button.

2 From the pop-up menu, choose Select Any. The Component Selection Criteria dialog box appears.

3 Enter the group number, as assigned at the schematic level, in the Group Number text box and choose the OK button. The group of components snaps to the cursor.

4 Click the left mouse button to place the components on the board.

Minimizing connections to optimize placement

Use the Minimize Connections command to evaluate the connections within a net and find the shortest route for the net (ratsnest) based on the placement of the pins or components on the board. When nothing is selected, Minimize Connections is a global command; it affects the entire board each time you apply it. However, if you have selected one or more components, Minimize Connections only affects the nets attached to the selected components. You can also select just a single net and minimize the connection length on that net only.

To use the Minimize Connections command

1. Choose the component toolbar button.
2. If desired, select the appropriate component(s) or net(s).
3. From the pop-up menu, choose Minimize Connections.

Copying, moving, and deleting components

You can copy components using the Copy command and delete them using the Delete command. You can switch between move mode and edit mode using the Move On/Off command. When you select a component, you can immediately begin moving it. If you choose Move On/Off, the component remains selected but freezes in place and can only be moved using the arrow keys. If you select a component using CTRL+left mouse button or SHIFT + spacebar, it remains stationary until you drag the cursor while pressing the left mouse button, or until you press an arrow key.

To copy a component

1. Choose the component toolbar button.
2. From the pop-up menu, choose Copy. A copy of the component attaches to the cursor.
3. Click the left mouse button to place the component.

To move a component

1. Choose the component toolbar button.
2. From the pop-up menu, choose Move On/Off. The component is highlighted, but remains in place.
3. Press the CTRL key and click the left mouse button to move the component.

To delete a component

1. Choose the component toolbar button.
2. From the pop-up menu, choose Delete. A dialog box asking you to confirm your decision to delete appears.
3. Choose the OK button. The component is deleted.

Swapping components

Use the Swap command to exchange the positions of two selected components.

To swap components

1. Press the CTRL key and select two components.
2. From the pop-up menu, choose Swap. The selected components switch places.

Rotating components

The Rotate command rotates any selected components around the lower left corner of the component (or component area, if you select more than one component), based on the Increment setting in the System Settings dialog box. The relationships between the components you select remain the same. The entire group rotates around the lower left corner, rather than each component rotating in its place.

To rotate components

1 Select one or more components.

2 From the pop-up menu, choose Rotate. The selected items rotate.

Note: To change the rotation increment, choose System Settings from the Options menu, then enter the number of degrees you want the components to rotate in the Increment text box in the System Settings dialog box. The default rotation increment is 90˚. You can also set a rotation increment to minute precision by typing the degrees of rotation followed by a space, followed by the number of minutes. You generally want to make the increment divisible by 360˚, so that the component returns to 0˚ rotation when it comes fully around.

Mirroring components

The Opposite command mirrors the components you have selected in the X dimension to the opposite side of the board.

To mirror components

1 Select one or more components.

2 From the pop-up menu, choose Opposite. The components are mirrored to the other side of the board.

Mirrored footprints in Capture and Layout

Layout v7.1x and later uses a user property called COMPSIDE to indicate which side of the board a component is on. By setting COMPSIDE=BOTTOM (or BOT), you can place a component on the bottom side of the board. COMPSIDE=TOP is the default.

Layout v6.42 did not have the COMPSIDE user property. It used a -M suffix to indicate a component that was mirrored to the bottom side of a board. Existing schematics/boards will work properly with the -M suffix, but for parts that have it, there

will not be an actual footprint that exactly matches the name found in Capture and Layout.

To eliminate the -M property, search the user properties (in Capture) for footprints that contain a -M suffix. Remove the suffix and add the COMPSIDE user property, then forward annotate to Layout. You can also remove the -M attribute in Layout by using the Footprints spreadsheet.

Related topics

Mirroring obstacles

Placing components using a matrix

You can place components using a matrix. Matrix placement is useful for placing groups such as memory arrays and discrete components. You can create a matrix of any size anywhere on the board. Then you can place a group of components into the matrix using the Matrix Place command (on the pop-up menu).

To place components using a matrix

1. From the Tool menu, choose Matrix, then choose Select Tool.
2. Place the pointer at the desired location for upper left corner of the matrix and, pressing the left mouse button, drag the mouse to the desired lower right corner and click the left mouse button.
3. Move the pointer up and down or left and right within the matrix, to create the desired number of cells. Click the left mouse button to stop drawing.
4. Choose the component toolbar button.
5. Select a group of components to place in the matrix.
6. From the pop-up menu, choose Matrix Place. The components are placed into the matrix.

To move a matrix line

1. From the Tool menu, choose Matrix, then choose Select Tool.
2. Click the left mouse button on any matrix line and move the mouse up or down for horizontal lines, or left or right for vertical lines.

To add a new line to a matrix

1. From the Tool menu, choose Matrix, then choose Select Tool.
2. Click the left mouse button on any matrix line and press the INSERT key to create a new line of the same type (horizontal or vertical).

To delete a line from a matrix

1. From the Tool menu, choose Matrix, then choose Select Tool.
2. Click the left mouse on any matrix line and press the DELETE key.

To move an entire matrix

1. From the Tool menu, choose Matrix, then choose Select Tool.
2. If you area-select the matrix, you can move the entire structure in any direction.

To copy a matrix

1. From the Tool menu, choose Matrix, then choose Select Tool.
2. If you area-select the matrix and press the INSERT key, you create a new, identical matrix.

Editing components

You can edit the component name, the footprint name, create mirrored components, lock or fix components, and enable or disable components for placement using the Edit Component dialog box.

To edit components

1 Select one or more components.

2 From the pop-up menu, choose Properties. The Edit Component dialog box appears.

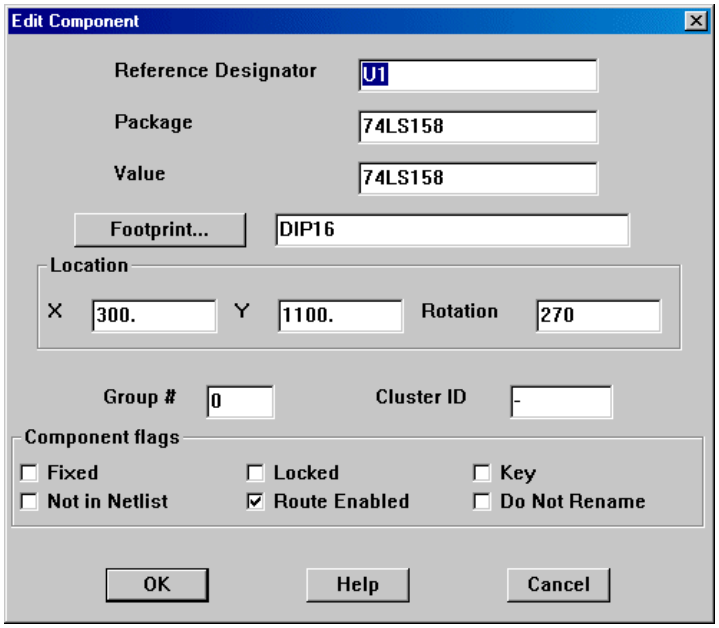


Figure 9-1 Edit Component dialog box

3 Edit the dialog box options as desired, then choose the OK button.

Reference Designator

The reference designator can be changed at any time (up to 100 characters are allowed). Layout remembers an infinite chain of name changes for back annotation purposes.

Package

Assigns an electrical package, including gate and pin swap information.

Value

Assigns a value to the component.

Footprint

Displays the Select Footprint dialog box, in which you can assign a footprint to the component.

X and Y

The X and Y text boxes contain the coordinates of the component's origin, relative to the board's (0, 0) origin. The coordinates are displayed in the units of measurement (mils, inches, microns, millimeters, or centimeters) that you selected in the System Settings dialog box (available when you choose System Settings from the Options menu).

Rotation

The rotation of a component can be specified in degrees (0 to 360) and minutes (0 to 60) of rotation from the origin of the component's associated footprint. If you type an integer without a suffix, Layout assumes it is in degrees. If you type two integers separated by a space, Layout assumes the second integer to be minutes of rotation. A single quote is optional, to indicate that the second integer represents minutes.

Note: If you are going to be rotating many components at odd angles, you can use the Increment text box in the System Settings dialog box (accessed by choosing System Settings from the Options menu) to set the rotation increment globally, so that you can use the

Rotate command on the pop-up menu to rotate components.

Group

A group number (0 to 100) is a permanent way of organizing components and helping Layout recognize which components should be grouped together, regardless of the phase of the design process. Typically, the component group number comes directly from the schematic input.

Cluster ID

Assigns the reference designator of the key component in a cluster as the cluster name for easy reference.

Fixed

Fixed components are permanently placed in a given location. The fixed designation can only be overridden by selecting the fixed component on the board, opening the Edit Component dialog box, and deselecting the Fixed option.

Not in Netlist

A component that does not display on the schematic is not in the netlist. If a component, such as a mounting hole, appears on the board but not on the schematic, and you do not want the component to be deleted when you run AutoECO, you must designate it as Not in Netlist.

Locked

Locked components are temporarily placed in a given location. The locked designation can be overridden by selecting the locked component, then choosing the OK button when a dialog box with the message “One or more components locked. Override?” appears. You can also override the locked designation by selecting the locked

component, opening the Edit Component dialog box, and deselecting the Locked option.

Route Enabled

If selected, tracks can be routed out of the component.

Key

Assigns a key component around which other associated components are placed.

Do Not Rename

Prevents the component from being renamed when you run the Rename Components command.

Selecting an alternate footprint

You can use the Select Footprint dialog box (accessed by choosing the Footprint button in the Edit Component dialog box) to change a component's footprint. When you do so, the previous footprint becomes an alternate, which is then listed in the Footprint Selection dialog box for that component.

You can use the Alternate Footprint command (on the pop-up menu) to select alternate footprints for components on the board. You may want to select an alternate footprint for a component if you are changing component technology. For example, you may want to replace a through-hole part with an SMT part.

To change a component's footprint

1. Choose the component toolbar button.
2. Double-click on a component. The Edit Component dialog box appears.

3 Choose the Footprint button. The Select Footprint dialog box appears.

4 In the Libraries list, select the library from which you want to select a footprint.

If the library is not displayed in the Libraries list, choose the Add button to add the library. If you do not know the name of the library that contains the required footprint, choose the Search button to search for the required footprint using the Search Footprint dialog box.

5 In the Footprints list, select a footprint. The footprint appears in the preview window.

6 Choose the OK button twice to close the dialog boxes. The footprint for the component is replaced. The alternate footprint is now available for selection in the Footprint Selection dialog box, as explained previously.

To select an alternate footprint

1 Choose the component toolbar button.

2 Select a component.

3 From the pop-up menu, choose Alternate Footprint. The Footprint Selection dialog box appears.

4 Select the desired alternate footprint and choose the OK button.

Merging .MAX files into a design

Merging .MAX files is most useful for placing components and merging the routing of a master board section into a design.

All of the information necessary to duplicate a section or template from one design to another is merged: grids, spacing, padstacks, package, footprints, components, nets, tracks, copper, free track, detail (such as obstacles), and text.

The Load Merge board command topic lists conventions used when a file is merged that you should be familiar with before using this command.

To merge a .MAX file into a currently open design

1. From the File menu, choose Load.
2. In the Files of type box, select Merge board (.MAX) Template (.TCH or .TPL).
3. Select the .MAX file or template file to be merged.

You are prompted: "If any components on the board you're loading have the same reference designator as an existing component, do you want to make a copy or update the existing comp? Answer YES for copy. Answer NO for update."

If you answer Yes, components in the merging file with the same reference designators as components in the open design are added to the open design, with three zeros appended to their names. For instance: U1 becomes U1000, R34 becomes R34000. The original components in the open design retain their reference designators.

If you answer No, components in the open design are replaced by components with the same reference designator from the merging file.

Related topics

Load Merge board command

Using autoplacement

Before you begin autoplacement, it is important to set up the board properly. Use the list below as a preplacement checklist, and ensure that these tasks are completed before you begin autoplacement.

- Check the board, place, and insertion outlines
- Check the place grid
- Check mirror layer or library layer settings
- Weight and color-code nets
- Check gate and pin data

- Check preplaced components and secure them on the board using the Lock or Fix commands
- Create component height keepins and keepouts, or group keepins and keepouts

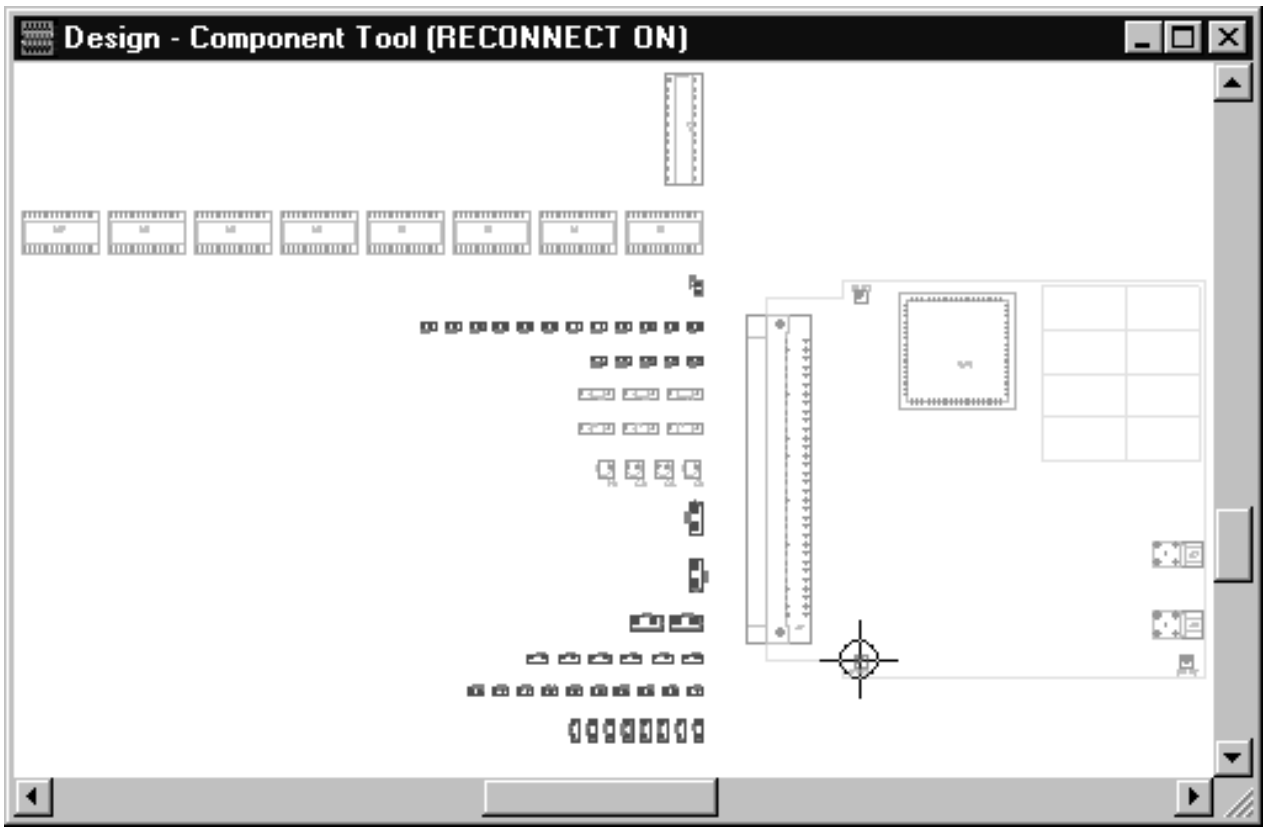


Figure 9-2 A board prior to component placement.

Once you have set up a board, you can begin placing components. You can have Layout Plus autoplace the entire board for you, you can place components in groups (using clusters), or you can place components individually.

- Autoplacement allows for a wide range of board complexity. By having Layout Plus autoplace the entire board, you can leave considerations about density and design rules to Layout Plus. This component placement

method is especially suited for situations that call for an analysis of all possible board configurations.

- Grouping components in clusters lends a logic and order to the design flow of your board. Grouping with clusters minimizes connection lengths and guarantees that similar components are grouped together.
- Layout Plus makes individual component placement easier with push-and-shove technology. Activating this technology (by choosing the Shove command) moves placed components out of your way, in order to make room for the component you're trying to place.

Placing components using autoplacement

Disabling power and ground nets

During autoplacement, you are primarily concerned with connectivity between components. Because nets on plane layers are largely irrelevant to the placement process, you can disable these nets for routing. This also significantly improves system performance during placement, since disabled nets are not redrawn.

To disable nets attached to plane layers

1. Choose the spreadsheet toolbar button, then choose Nets. The Nets spreadsheet displays.
2. Using the CTRL key and the left mouse button, select all the cells in the Routing Enabled column that both pertain to nets attached to plane layers and read "Yes." Release the CTRL key.
3. From the pop-up menu, choose Enable<->Disable. The entries for the cells you selected change from "Yes" to "No," indicating that the nets are disabled.

Autoplacing components

Once the board is set up properly and you have loaded a strategy file, you can choose the Board command to automatically place all of your components. When you choose this command, Layout Plus completes six passes of component placement. The progress of the operation displays in the status bar at the bottom of your screen.

To autoplace components

1 From the Auto menu, choose Place, then choose Board.

Pass 0

Performs an initial Proximity Place pass that clusters components based on interconnectivity and then places those clusters in locations that favor shorter connections. Pass 0 uses a minimal number of iterations (repeated algorithms) and attempts (different placements attempted).

Pass 1

Performs an Assign Clusters pass that takes all of the components that are not locked or fixed and puts them in clusters according to their interconnectivity and whether the components are grouped or not.

Pass 2

Performs a Place Clusters pass that places the clusters on the board based on connectivity and their position relative to other clusters or fixed components.

Pass 3

Performs a Proximity Place pass that uses a larger number of iterations and attempts than were used in Pass 0. This process places components more accurately.

Pass 4

Performs a Swap Comps pass that swaps neighboring components to see if placement can be improved by reducing connection length and crossovers.

Pass 5

Performs an Adjust Comps pass that adjusts the components in order to eliminate any overlapping.

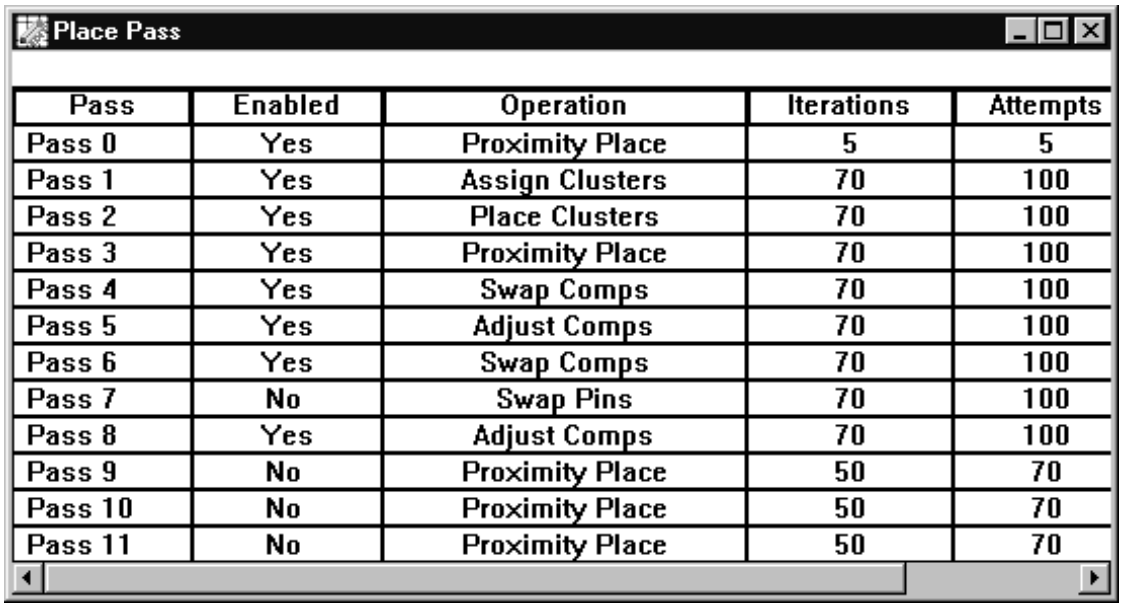
Note: After you place the board, you can view component placement statistics using the Statistics spreadsheet. For information on viewing component placement statistics, see Viewing placement statistics in Chapter 7: Placing and editing components in the OrCAD Layout User's Guide.

Editing place pass information

Layout Plus can also perform additional passes (Pass 6 through Pass 11). Though these are disabled by default, you can enable them in the Place Pass spreadsheet. You can also alter the number of iterations and attempts performed by the six standard passes, as described above.

To edit place pass information

1 Choose the spreadsheet toolbar button, choose Strategy, then choose Place Pass. The Place Pass spreadsheet displays.



Pass	Enabled	Operation	Iterations	Attempts
Pass 0	Yes	Proximity Place	5	5
Pass 1	Yes	Assign Clusters	70	100
Pass 2	Yes	Place Clusters	70	100
Pass 3	Yes	Proximity Place	70	100
Pass 4	Yes	Swap Comps	70	100
Pass 5	Yes	Adjust Comps	70	100
Pass 6	Yes	Swap Comps	70	100
Pass 7	No	Swap Pins	70	100
Pass 8	Yes	Adjust Comps	70	100
Pass 9	No	Proximity Place	50	70
Pass 10	No	Proximity Place	50	70
Pass 11	No	Proximity Place	50	70

2 Double-click in the Pass cell that corresponds to the pass you want to modify. The Edit Place Pass dialog box displays.

3 Change the settings as desired, then choose the OK button.

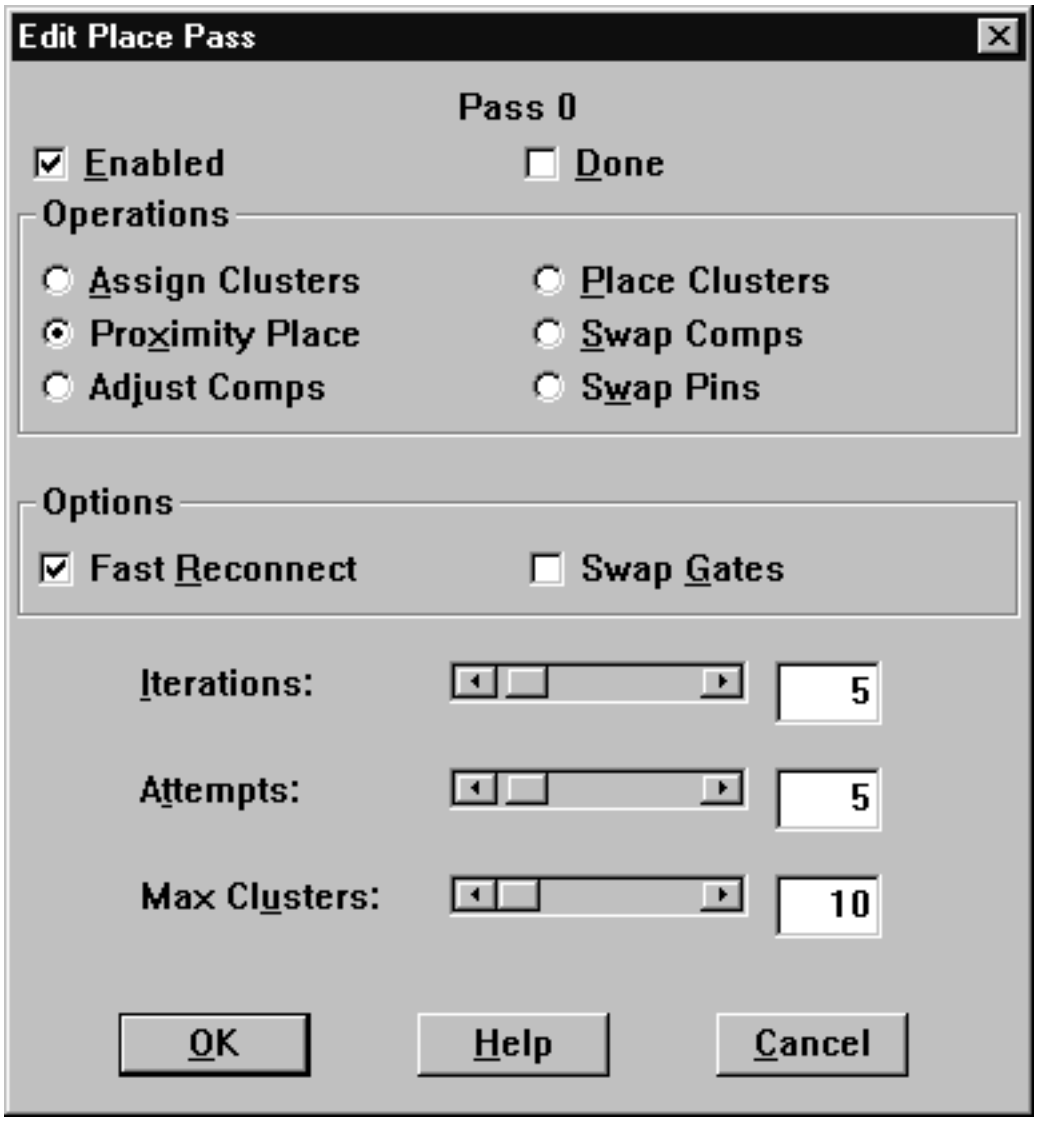


Figure 9-3 The Edit Place Pass dialog box

Enabled

Enable passes by selecting the Enabled option. Each pass displays either a Yes or a No in the Enabled column in the Place Pass spreadsheet, indicating that the Enabled option is either selected (Yes) or not selected (No) for the pass. If you

decide that a pass is not necessary, you can deselect the Enabled option for the pass, in which case the autoplacement routine does not run the pass.

Done

A pass is automatically set to Done upon completion. This indicates which passes have been completed in the event that the autoplacement routine is interrupted. Once restarted, the autoplacement routine does not run any passes that are marked as Done.

Note: You can use the Done option to temporarily disable a pass, while leaving the underlying Enabled setting intact. This offers the advantage that you need only deselect the Done option, and everything will be returned to its previous state (either enabled or not enabled)

Assign Clusters

Automatically groups components according to connectivity before they are placed on the board. Clusters maximize the number of connections between components within the same cluster, while minimizing the number of connections between clusters.

Proximity Place

Uses cluster locations as a starting point for board placement, then considers thousands of design permutations in order to choose a placement that optimizes quality.

Adjust Comps

After the clusters are broken, Layout Plus adjusts individual components in a rough grid pattern for a neater, more efficient placement.

Place Clusters

Places the assigned clusters in the best location on the board, relative to other clusters and fixed components.

Swap Comps

Swaps adjacent components in order to refine a nearly completed placement.

Swap Pins

Swaps pins using the package information from the schematic, to allow automatic pin swapping during autoplacement. This is not allowed if you created your board without a schematic.

Fast Reconnect

Both the Proximity Place and the Swap Comps passes automatically use this option. You can disable this option to achieve a slightly better quality placement, but at lesser speed.

Swap Gates

Swaps gates in order to reduce the overall connection length on the board.

Iterations

Controls the number of algorithms Layout Plus employs during a given placement. The higher the number, the more algorithms Layout Plus uses. A low number results in a faster placement, but a decreased chance that Layout Plus will find the right placement for the board.

Attempts

Determines how many placements are attempted during each iteration. The higher the number, the more placement attempts are made, and the better the chance that Layout Plus finds the optimum placement for the board.

Max Clusters

Specifies the maximum number of clusters that Layout Plus uses during autoplacement. It is probably best to use the default set by your placement strategy file for maximum clusters. If you choose too high of a value, autoplacement spends a great deal of time forming the clusters.

Using interactive placement commands

Layout Plus provides commands you can use to optimize placement. Using these commands, you can hide routes and connections, and shove, cluster, and adjust components to control exactly where they are placed on the board.

Hiding routes and connections

Reconnection mode prevents routes and connections from displaying on the screen, making it easier for you to see components and place them. With reconnection mode enabled, only nets connected to components you select are visible. As you move a component, its associated nets move with it, allowing you to account for connectivity while placing a component.

Note: If you are working with large nets, you can choose Minimize Connections from the pop-up menu while you're moving a component. The shortest route for signals associated with the component is shown.

To hide routes and connections

1 Choose the reconnect toolbar button.

Or

1 From the Options menu, choose User Preferences. The User Preferences dialog box displays.

2 Select the Instantaneous Reconnection Mode option, then choose the OK button.

The reconnect toolbar button is “pushed in” and the phrase RECONNECT ON is added to the design window’s title bar.

To re-display routes and connections

1 Choose the reconnect toolbar button.

Or

1 From the Options menu, choose User Preferences. The User Preferences dialog box displays.

2 Deselect the Instantaneous Reconnection Mode option, then choose the OK button.

The reconnect toolbar button is not “pushed in” and the phrase RECONNECT ON is removed from the design window’s title bar.

Shoving components

You can use the Shove command to automatically move previously placed components so that a component you have selected and are trying to place has enough room to be placed on the board. Layout Plus uses place outlines and insertion outlines to regulate this process. Insertion outlines can overlap each other, but place outlines cannot. The direction a component is shoved is determined by the degree of overlap between the component you are trying to place and the previously placed components.

To shove components

1 Choose the component toolbar button.

2 Select a component for placement. The component attaches to your pointer.

3 As you are positioning the component, choose Shove from the pop-up menu. Layout Plus moves other components away from the component being placed.

Adjusting components

You can use the Adjust command to line up components, based on their connectivity.

Note: The components that you want to align should be in close proximity. This command is helpful for precise (not general) placement.

To adjust components

1 Choose the component toolbar button.

2 Press CTRL and select two components that you want to adjust.

3 From the pop-up menu, choose Adjust. Layout Plus aligns the components.

Placing components using clusters

Note: When reconnection mode is enabled, a cluster is represented by a circle. When reconnection mode is disabled, the components are individually drawn; you need only click on one component to select them all.

Clusters are component groups formed to simplify placement. As with groups, clusters allow you to move multiple components at once, can represent specific circuits, and can be placed quickly in the appropriate area of the board. Compared to a number of individual components, clusters are simple from a graphic standpoint, which makes them ideal for quickly testing various placements, since your system can redraw clusters rapidly.

When reconnection mode is enabled, a circular border represents the combined area of all of the components within

a cluster. Use the circle as a visual aid that indicates the amount of room needed to place the cluster in an area of the board.

Note: If you cannot see a cluster's circular border, your place outlines are either defined as invisible, or they are a color that is incompatible with the background color. To see a cluster's border clearly, choose the color toolbar button. In the Color spreadsheet, select the row(s) for Place outline using the CTRL key and the left mouse button. Choose Visible<->Invisible from the pop-up menu (when an entity is set to visible, its spreadsheet color displays as a solid color, not as a diagonal pattern). If the color of the place outline (the cluster's circular border) needs adjusting, assign a new color by double-clicking on the color cell for that row. A color palette displays. Select a color that will contrast well against the background color, then choose the OK button. Close the Color spreadsheet.

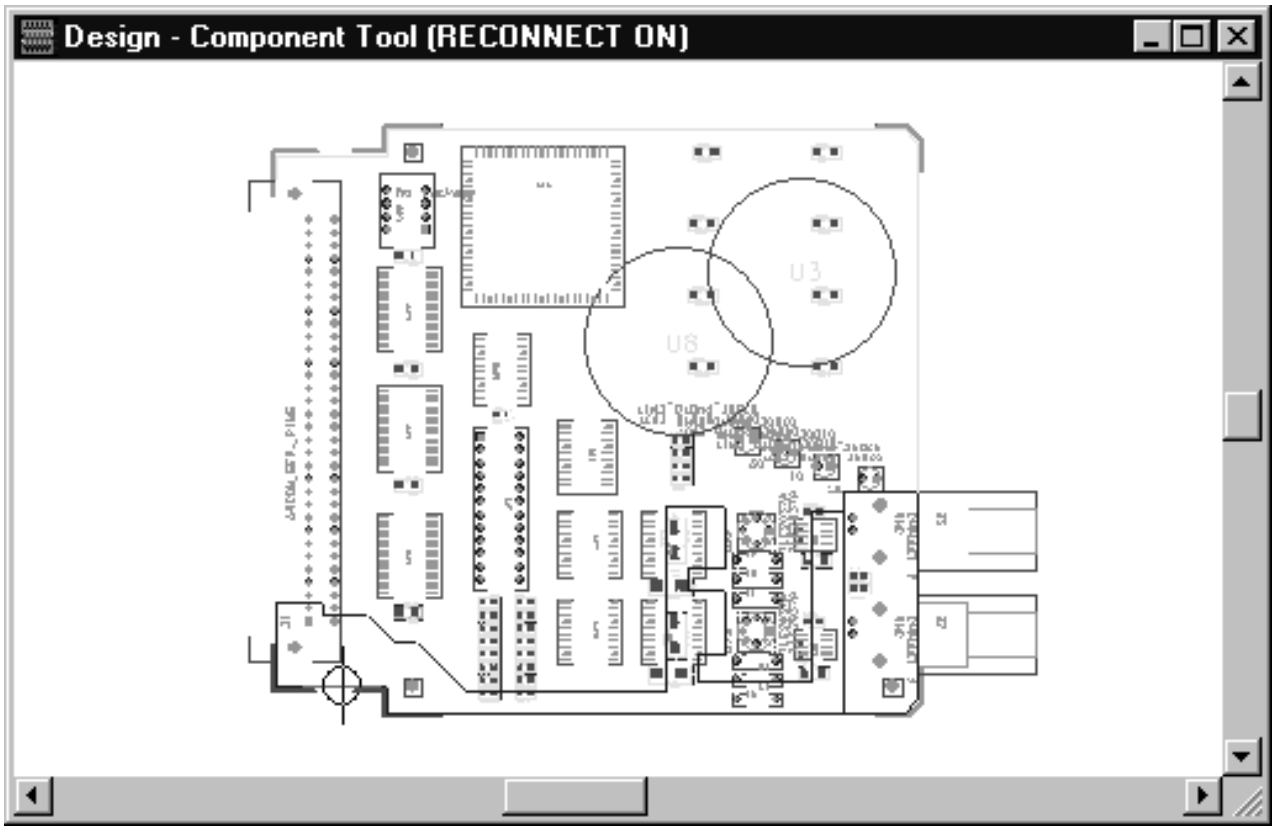


Figure 9-4 U3 and U8 are clusters.

To place components using clusters

1. Choose the component toolbar button.
2. From the pop-up menu, choose Select Any. The Component Selection Criteria dialog box displays.
3. Enter a group number in the Group Number text box and choose the OK button. You can create groups at the schematic level, or you can build them in Layout Plus.

   Note: To group components in Layout Plus, choose the spreadsheet toolbar button, then choose Components. Select the components that you want to group by pressing CTRL and clicking the left mouse button. From the pop-up menu, choose Properties to display the Edit Component dialog box. Type an integer in the Group # text box and choose the OK button.
4. From the pop-up menu, choose Make.
5. Position the cluster on the board and click the left mouse button to place it.

Breaking clusters

You can also use the Break command to dissolve a cluster into its individual components. Break unclusters the components, but does not change the location or parameters of any of the components within the cluster, nor does it separate the components from an assigned group.

To uncluster components

1. Choose the component toolbar button.
2. Select the cluster by pressing CTRL and clicking with the left mouse button.
3. From the pop-up menu, choose Break. The cluster is broken down into individual components.

Placing clustered components using Quick Place

After you place clusters on a board, Quick Place can unstack and arrange their components, separating them efficiently according to their place outlines.

Note: In Layout Plus, you create a place outline by creating an obstacle, and then defining it as a place outline. For information on doing this, see Creating obstacles on page 254.

To unpack and place clustered components using Quick Place

1. Choose the component toolbar button.
2. Select the cluster(s) by pressing CTRL and clicking with the left mouse button.
3. From the pop-up menu, choose Quick Place.

The individual components of the cluster(s) you selected are placed quickly, according to each component's place outline.

Using circular placement

Circular placement may begin with or without selecting a component. You use the Circular Placement dialog box to set up circular placement.

Note: You cannot select multiple components for circular placement. If you do select more than one component, Layout Plus chooses one from the selection to use for circular placement.

If a component is selected, its current footprint name, group number, location (relative distance from circle center), rotation, radius, start angle from (0,0), and component angle are included in the dialog box when you open it. If you change the values for location, rotation, radius, or angle, it causes the component to be moved or rotated. Changing the reference designator does not affect the selected component.

During circular placement, the board datum temporarily shifts to the center of the proposed circle or arc. The values in the dialog are calculated relative to this temporary datum.

When you enter values for certain options in the dialog box, Layout Plus calculates the effect of the values on other parameters within the dialog box. For example, consider a board with its circle center at 0, 0. If you set the circle's radius to 1000 mils and the Start Angle to 45 degrees, the Rel Start automatically calculates to 707.100, 707.100. These values display when you tab to another option in the dialog box, click in another field, or when you choose the OK button.

Note: For more information on how option values affect other option values, see *Auto-updating*.

There is no error checking available to prevent component overlap. Angular values must be positive or negative values within the range of 0 to 360 degrees. Real numbers are supported, as are degrees and minutes. For example, an angle of 45.5 degrees is equivalent to 45 degrees 30 minutes, and both values are supported.

The following dialog box values are preset, based on a selected component:

- Footprint Name
- Group Number
- Circle Radius
- Start Angle
- Rel Start X, Y
- Comp Angle

If a component is not pre-selected, all dialog box values are persistent upon re-invocation of the dialog box, with the exception of Ref Des. If a component is preselected, the following dialog values are persistent upon re-invocation of the dialog box:

- Comp Count
- Angle To Fill

- Angle Between
- Comp Angle Increment
- Added Comp Angle

Note: Blank fields are not legal.

To use circular placement

1. With no component selected, choose Circular Placement from the Auto menu. In the Circular Placement dialog box, choose the Footprint button, locate and select the footprint that you want to use for circular placement in the Select Footprint dialog box, then choose the OK button.
   or
   In the design window, select a component while pressing the CTRL key. From the Auto menu, choose Circular Placement. In the Circular Placement dialog box, the footprint name, group number, relative location to circle center, current rotation, current radius from circle center, current start angle from (0,0), and component angle are entered for you.

   Note: In the Select Footprint dialog box, select Local in the Libraries window to select a part from your current board file.

2. Enter new values as desired for the dialog box options. The options are automatically calculated and updated according to the inter-relationships of their values. See *Auto-updating* in this chapter for specific information.

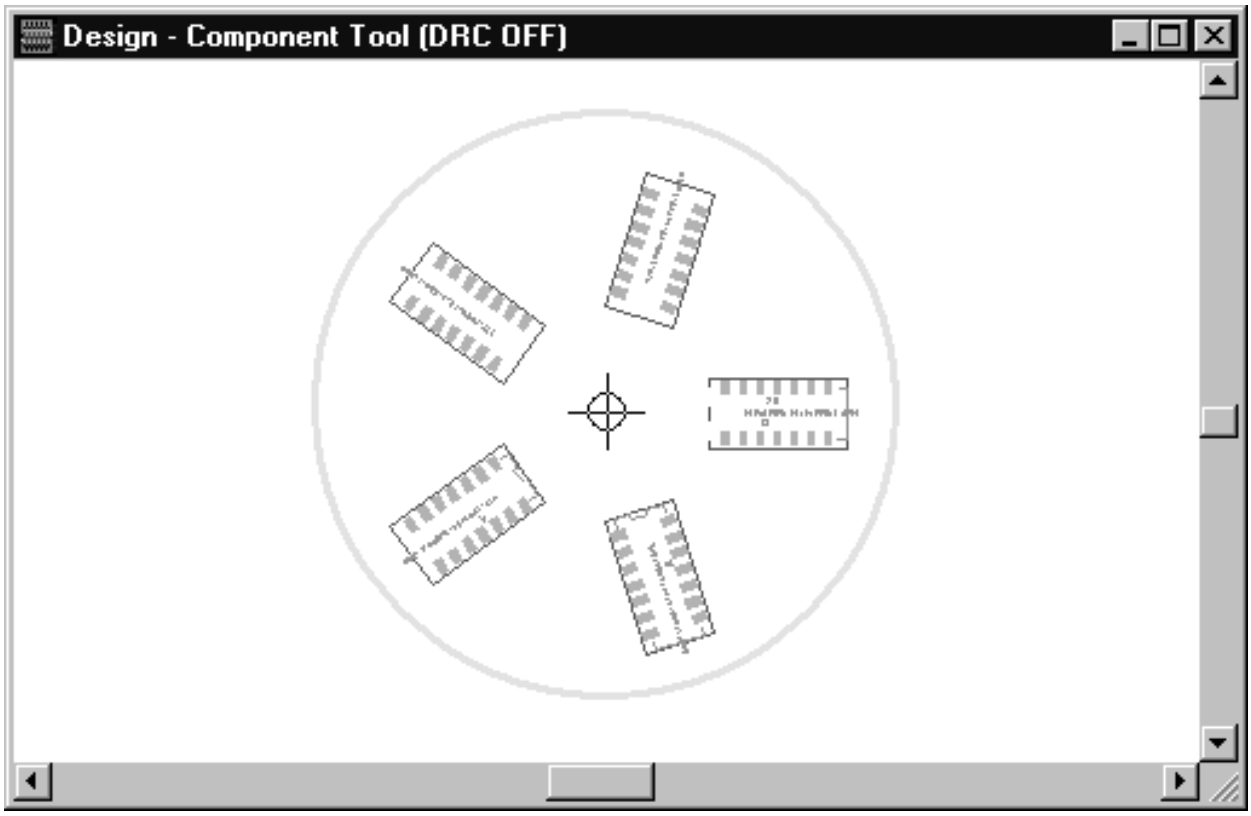
3. Enter the number of components that you want to place in the Comp Count text box and select the Use Angle to Fill or Use Angle Between option. The Comp Angle Increment is automatically calculated, based on these entries.

   Note: The Comp Count includes the selected component.

4. Choose the OK button. The components are placed according to the values specified.

   Note: Circular placement can be reversed using the

Undo command only immediately after placement, but not following any subsequent commands.



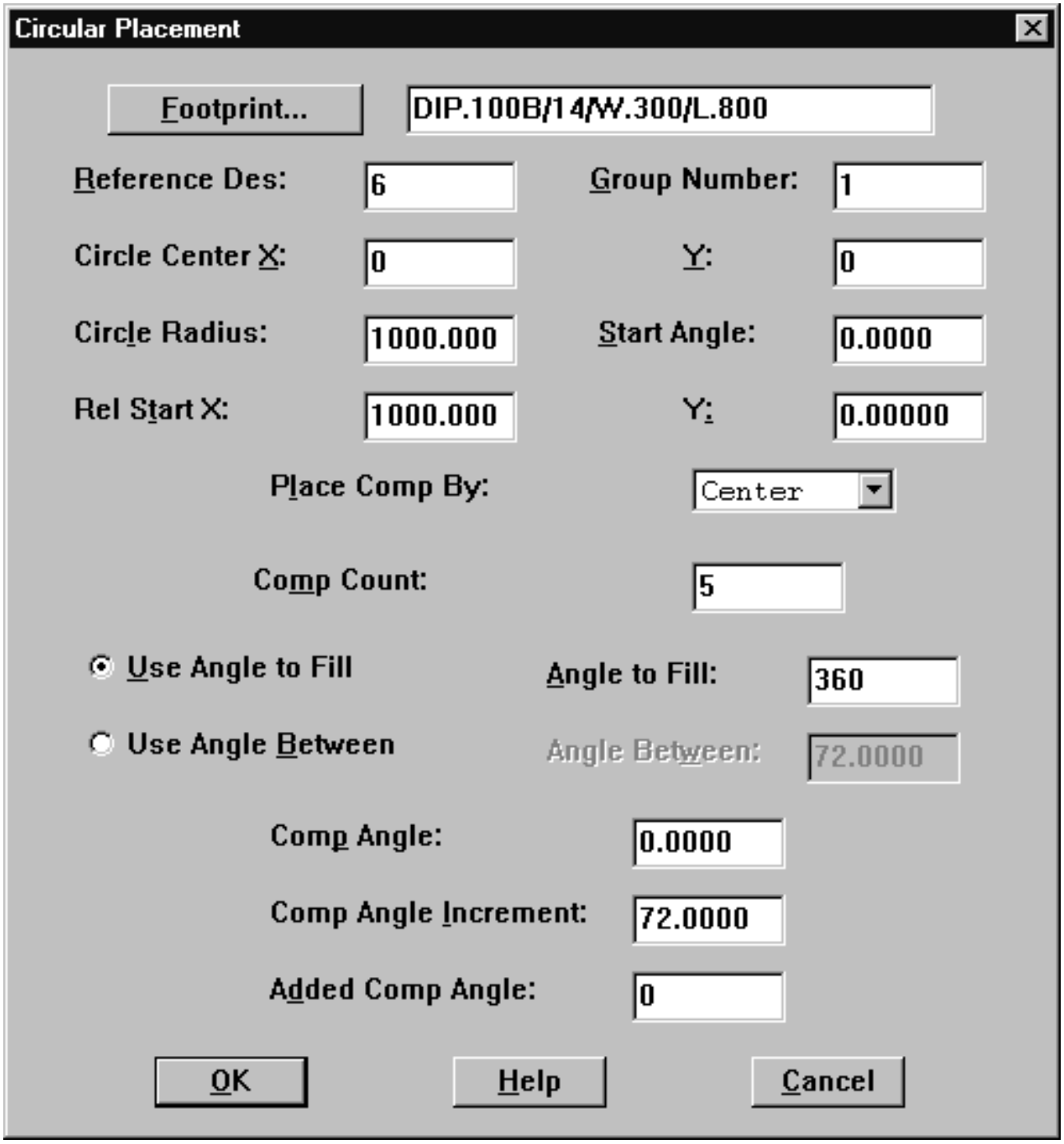


Figure 9-5 The Circular Placement dialog box

Footprint

Displays the Select Footprint dialog box, in which you can select a library and then a footprint for circular placement.

Reference Des

The reference designator for the selected component. Default shown is the next unused reference designator for the board. Or, you may enter your own unique reference designator.

Group Number

Group number to associate with added components. The default is 0 (zero), meaning components do not belong to any group.

Circle Center X, Y

Coordinates of the circle center.

Circle Radius

Radius of the circle of placed components. The radius is measured from the circle center to the component placement point. The component is placed with its origin on this placement point.

Start Angle

Beginning placement angle of first component added or selected.

Rel Start X, Y

Placement location for first component added. This value is measured as a relative distance from the Circle Center value. If a component has been selected on the board, changing

these values will cause the selected component to move and placement to begin from this new location.

Place Comp By

The drop-down list gives you choices of how the components are to be placed. Datum (the default) uses the board datum, Center uses the center of the component, Pin 1 uses pin 1 of the component, and Insert Pt uses the insertion point of the component. Regardless of how the component is placed, if it is subsequently rotated, it rotates about the datum.

Comp Count

The total number of components to be added, inclusive of a selected component.

Use Angle to Fill

Toggle button to select between Angle to Fill and Angle Between.

Angle to Fill

The angle to be filled by added components.

Use Angle Between

Toggle button to select between Angle to Fill and Angle Between.

Angle Between

The space or angle between each added component’s placement point.

Comp Angle

The rotational angle of each added component. Changing this value will cause a selected component to rotate.

Comp Angle Increment

A successive rotation angle increment for each added component, calculated before placement. This will not affect a selected component; however, each added component will be rotated by this increment. For example, starting at 0, a Comp Angle Increment value of 20 degrees would cause components to be rotated before placement at 0, 20, 40, 60, and so on.

Added Comp Angle

A rotational angle that is added to each component after it is placed. This command rotates the individual components in place, around their graphic origins. For example, starting at 0, if Comp Angle Increment is 20 and Added Comp Angle is 5, the component rotations would be 5, 25, 45, 65, and so on.

Auto-updating

Certain dialog box options are automatically updated to reflect how they are affected when other dialog box values are changed. The updated values display when the focus is changed in the dialog box using the TAB key, or by clicking the left mouse button on any other item in the dialog box. They are also updated when you choose the OK button to exit the dialog box. These relationships are listed below:

- Changing Circle Center X, Y automatically updates Circle Radius, Start Angle, Rel Start X, Y, and Comp Angle.
- Nothing automatically updates Circle Center X, Y.
- Changing Circle Radius automatically updates Rel Start X, Y.
- Changing Start Angle sets Comp Angle to the same value, and automatically updates Rel Start X, Y (by

default, Start Angle and Comp Angle should be the same).

- Changing Rel Start X, Y automatically updates Circle Radius, Start Angle, and Comp Angle.

  Note: By default, Start Angle and Comp Angle should be the same. However, you can change Comp Angle and it will not update any other option.

- Changing Comp Count automatically updates Angle to Fill, Angle Between, and Comp Angle Increment.
- Nothing automatically updates Comp Count.
- Changing Angle to Fill automatically updates Angle Between and Comp Angle Increment.
- Changing Angle Between sets Comp Angle Increment to the same value and automatically updates Angle to Fill.

  Note: By default, Angle Between and Comp Angle Increment should be the same. However, you can change Comp Angle Increment and it will not automatically update anything else.

Adding footprints to the board

If you add a spare component to the board, such as a mounting hole, or if you did not bring in a netlist and are therefore adding components to the board manually, you can use the Add Component dialog box to bring footprints onto the board.

Note: If you change a footprint on the board, be sure to back annotate the change to the schematic.

To add a component footprint to the board

1. Choose the component toolbar button.
2. From the pop-up menu, choose New. The Add Component dialog box appears.

3 Choose the Footprint button. The Select Footprint dialog box appears.

4 In the Libraries list, select the library from which you want to select a footprint.

If the library is not displayed in the Libraries list, choose the Add button to add the library. If you do not know the name of the library that contains the required footprint, choose the Search button to search for the required footprint using the Search Footprint dialog box.

5 In the Footprints list, select a footprint. The footprint appears in the preview window.

6 Choose the OK button twice to close the dialog boxes. The footprint is attached to the cursor.

7 Place the footprint in the desired location on the board by clicking the left mouse button.

Linking components to footprints

Two common errors that can occur during the AutoECO process when opening a design are:

- Mounting holes disappear from the board when you run AutoECO.
- The pin numbers from the schematic do not match the pad names in Layout.

Mounting holes disappear from the board when you run AutoECO

A mounting hole is non-electrical, not in the netlist, and is subject to being deleted when you run AutoECO. Because of that, you should check the Not in Netlist flag in the Edit Component dialog box.

To set the Not in Netlist flag

1 Choose the Spreadsheets toolbar button.

2 Select Components from the drop-down list.

3 Locate the component and double-click on it. The Edit Component dialog box displays.

4 Select the Not in Netlist check box.

5 Click OK.

Footprint pin names do not match schematic symbol pin numbers

Pin numbers in the schematic must match the footprint pin names in the footprint library files. For example, a diode in the schematic might have pins called Anode and Cathode, while the actual footprint has corresponding pin names of Ano and Cath, or 1 and 2. These differences must be reconciled or the design will not load. To correct this situation, either:

- change the symbol pin names in the schematic to match the footprint pin names in the Layout library.
- change the footprint pin names in the library to match the symbol pin names.

Each device in the schematic describes an electrical part. For example, a description could be 74LS00. Electrical parts are matched to footprints in one of three ways:

- The part contains a footprint attribute, such as DIP14, that matches a footprint found in the Layout footprint library.
- The part name 74LS00 is linked to a footprint in the SYSTEM.PRT file located in the LAYOUT\DATA subdirectory.
- If you are in the process of running AutoECO, and Layout is unable to find a designated footprint, the Link Footprint to Component dialog box appears.

Related topics

Alternate Footprint command (Component)

Components spreadsheet

Library Manager command (LSession: Tools menu)

Managing libraries and footprints

Checking placement

You should check the placement of a board using Placement Spacing Violations, the density graph, and the placement information in the Statistics spreadsheet.

Using Placement Spacing Violations

Before you route the board, you should run Placement Spacing Violations, which looks for component-to-component spacing violations and other placement errors, such as components that violate height restrictions, insertion outlines, or grid restrictions.

Note: Placement Spacing Violations uses component outlines to determine whether there is a spacing violation. Therefore, component outlines should encompass the entire area of the IC or discrete component, including such objects as pinout patterns and sockets.

Any problem found by Placement Spacing Violations is marked with a circle. You can determine the nature of the problem by choosing the query toolbar button, which brings up the query window. Then, when you choose the error toolbar button and select the error, the information about the error appears in the query window.

Note: For information on how to use the error tool to get more information about reported errors, see *Ensuring manufacturability*.

To check placement spacing violations

1 From the Auto menu, choose Design Rule Check. The Check Design Rules dialog box appears.

2 Choose the Clear All button.

3 Select the Placement Spacing Violations option, then choose the OK button. Layout checks the board for component placement violations and marks any errors with circles.

Using the density graph

The density graph displays a graphical representation of the connection density of your board. Using colors ranging from blue and green (acceptable density) to pink and red (very dense), the density graph represents the degree of difficulty that will be faced in routing the board.

The density graph analyzes all routing layers, routed tracks, widths of tracks, spacing rules, DRC settings, and connections to calculate the available routing channels. It shows the crossing count at each location of the board in relation to how much of each cell is being filled by a pad, track, or connection.

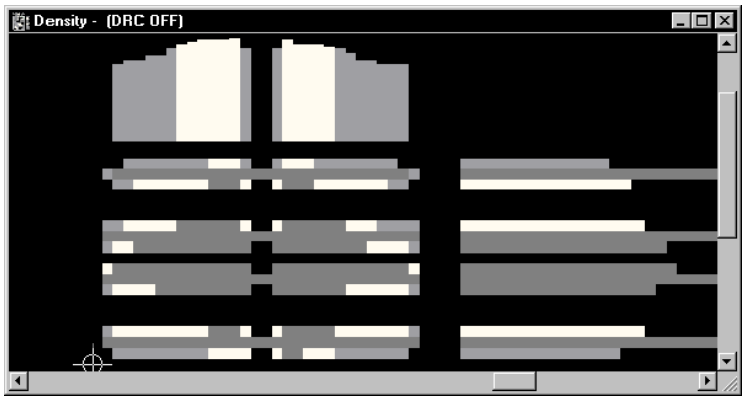
There are two kinds of data shown on the density graph: the *board density* at each location (the number of pads and connections in a given area of the board), and the *track density* (the track density in each channel), shown as bar graphs at the top and right.

To open the density graph

1 From the View menu, choose Density Graph. The density graph window appears.

Note: A small amount of red in the density graph is acceptable, but you should attempt to keep the percentage of red below 25%, because a board that is more than 25% red is likely to encounter serious routing

difficulties.



2 To return to the design window, choose Design from the View menu.

Viewing placement statistics

When you finish placing components on the board, you can view the component placement statistics in the Statistics spreadsheet. The spreadsheet shows the percentage and number of components placed, how many were placed off the board, how many were unplaced, and how many were placed in clusters.

To view placement statistics

1 Choose the spreadsheet toolbar button, then choose Statistics. The Statistics spreadsheet appears.

2 Scroll until you find the % Placed row, which is the beginning of the placement data.

3 Close the spreadsheet when you are finished viewing the statistics.

Routing the board

10

After you have placed the components, you can route the board to form the electrical connections between the components. This chapter explains how to route the board manually, and describes the manual routing tools.

You can route the entire board manually using the routing tools described in this chapter. Or, if you have purchased Layout or Layout Plus, you can use the autorouter and interactive routing tools to route the board, then use the manual routing tools described in this chapter to optimize routing.

You probably performed the following tasks when you set up the board and placed components. If not, you need to do so to prepare the board for routing.

- Designate appropriate layers as plane layers or routing layers

  Note: For information on designating plane layers, defining vias, and setting net properties, see *Chapter 5, "Setting up the board."* For information on running Placement Spacing Violations, see *Checking placement*.

- Define vias
- Set or verify net properties
- Run Placement Spacing Violations and correct any spacing violations

After you have completed the above items, you are ready to begin the routing process. The steps in the manual routing process are:

- Check the board outline, via definitions, and routing and via grids
- Load a routing strategy file
- Route power and ground
- Fan out SMDs and verify connections to power and ground
- Route the remaining signals using the manual routing tools
- Optimize routing using the manual routing commands
- Check for route spacing violations and check routing statistics

Routing the board manually

When you view the board before you've done any routing, you'll see that the parts have many fine lines running between them. These lines are known as a *ratsnest*. A ratsnest represents the connections that need to be routed to form the necessary tracks on the board.

Note: Yellow triangles in a ratsnest indicate unrouted, zero-length connections (connections that lead directly from a pad on the top layer to a pad on the bottom layer without traveling in the X or Y direction).

A connection is an electrical path between two pins: a ratsnest represents an unrouted connection, while a track represents a routed connection.

Checking the board outline, via definitions, and routing and via grids

Before you route, you should check the settings for the board outline, vias, routing grid, and via grid.

- Verify that the board outline has a desirable amount of internal clearance, that there is only one board outline, and that it is on the global layer.
- Inspect the vias in the Padstacks spreadsheet to make sure that they are the right size and on the correct layers.
- Verify that the routing grid and via grid match for the placement of tracks.

Note: For information on creating and editing a board outline, defining vias, and setting the routing and via grids, see *Chapter 5, "Setting up the board."*

Loading and editing a routing strategy file

Routing strategy files determine the parameters that guide routing, such as which default routing layers to use, when to use vias, in which direction the track should travel, and the size of the active routing window. They also set up the appropriate graphical display for routing.

The parameters for the strategy files provided with Layout are set according to the type of board (component type, number of layers) for which the strategy file is intended. In most circumstances, these parameters do not need to be changed. Changing the parameters can negatively affect routing results.

The strategy files are based on three sets of criteria: sweep, pass, and layer. To adjust for special circumstances before autorouting, you may need to modify the routing strategy files using the Route Sweep spreadsheet, Route Pass spreadsheet, and Route Layer spreadsheet.

When you select Strategy from the Spreadsheets toolbar button drop-down list, you can access these spreadsheets. By editing the parameters in these spreadsheets, you can create new strategy files (.SF) and edit the existing strategy files that drive autorouting. The spreadsheets are described briefly below.

- From the Route Sweep spreadsheet, you can access the Sweep Edit dialog box in which you can edit the route sweep parameters set by your strategy file. The dialog

box appears when you double-click on any cell in the spreadsheet, or when you select a cell and choose Properties from the pop-up menu.

- During autorouting, the router makes a specified number of cycles or *sweeps* through the current working area (component, window, or board). During the sweeps, the autorouter uses different routing mechanisms in the attempt to route the connections. Using the Route Pass spreadsheet, you can access the Edit Route Pass dialog box in which you can enable and disable the sweeps and sweep passes set by your strategy file. The dialog box appears when you double-click on any cell in the spreadsheet, or when you select a cell and choose Properties from the pop-up menu.
- The Route Layer spreadsheet contains information such as the *primary direction* in which tracks should travel on a given layer (horizontal or vertical), *between pins cost*, and *layer cost*. Using the Route Layer spreadsheet, you can access the Edit Layer Strategy dialog box in which you can edit the route layer parameters set by your strategy file. The dialog box appears when you double-click on any cell in the spreadsheet, or when you select a cell and choose Properties from the pop-up menu.

There are many routing strategy files provided with Layout, among which are files for two-layer, four-layer, six-layer, and eight-layer boards. You can load the routing strategy file that is most suitable for your board.

Note: For a complete list of the routing strategy files provided with Layout, see *Routing strategy files*.

To load a routing strategy file

1. From the File menu, choose Load. The Load File dialog box appears.
2. If necessary, change Files of type to Strategy.
3. Select a routing strategy file (.SF), then choose the OK button.

Note: Layout provides two types of strategy files: placement strategy files and routing strategy files. Although both types of files have an .SF extension, placement strategy files begin with the letters “PL.”

Before you load a new routing strategy file, ensure that the order of layers in the strategy file and the board are the same. If the order of layers in the strategy file and the board are not the same, the routing information in the strategy file will not be correctly loaded on the board and you will have to manually correct the strategy information.

Changing board density using routing strategy files

Note: For information about opening and viewing the density graph, see *Using the density graph*.

If your board is too dense in certain areas (indicated as dark red in the density graph), you can improve the density by experimenting with different routing strategy files or changing the placement. For example, you may want to add layers or change track width or track spacing rules.

To experiment with different routing strategy file

1. With the density graph window displayed, from the File menu, choose Load. The Load File dialog box appears.
2. Locate and select a strategy file (.SF), then choose the Open button. The density graph redraws itself, presenting new board density data resulting from loading the strategy file.

Routing power and ground

The steps in the power and ground routing process are:

- Enable the power and ground nets for routing and disable the other nets
- Perform fanout to connect SMDs to the plane layers
- Verify proper connection to the plane layers for through-hole components
- Disable the power and ground nets for routing and enable the remaining nets

Plane layers are typically used for power (VCC) and ground (GND). When routing multilayer boards, it is essential to route power and ground first. To do so, you enable the power and ground nets for routing, while disabling all the other signals for routing. After routing power and ground nets, you must disable them and enable all other signals for routing. Then you can route the remaining signals.

Note: Before you can route power and ground, you need to designate plane layers in the layer stack. For information on designating layers as plane layers, see *Defining the layer stack* on page 238.

Note: If you're routing nets with thousands of pins, you can disable Layout's default dynamic reconnect method, which is a method of calculating where the closest pin belonging to the same net you're routing is, then redrawing the ratsnest line to connect to the closest pin. Selecting the No Dyn Reconn option disables dynamic reconnect, with the result that you don't have to wait for Layout's ratsnest calculations and redrawing. Follow steps 5 though 8 in To manually route nets with planes and copper pours on the next page.

On surface mount technology (SMT) boards, you should fanout the board with only the power net enabled, to connect surface mount devices (SMDs) to the plane layers.

On through-hole boards, the appropriate nets are automatically attached to the plane layers with thermal reliefs. If the power or ground nets did not connect to the plane layers, one of three errors may have occurred in the netlist:

- The global power pin is not defined in the part.

- The pin is not connected to the proper signal.
- If the pin is connected, it does not have the correct signal name.

To remedy the problem, either modify the schematic and annotate it again, or modify the board by adding a pin to the signal. Keep in mind that this board modification cannot be back annotated to the schematic.

Note: For information on adding pins to nets, see *Adding and deleting pins connected to nets* on page 431.

Connections to the planes can be verified prior to post processing by verifying that only nets connected to the planes are enabled, then viewing the Statistics spreadsheet to verify that these nets are 100% routed.

Note: You can also view the thermal connections using the post process preview. For more information, see Previewing thermal reliefs.

When you are manually routing a net whose connectivity is partially satisfied by planes and copper pours, you may find it easier to follow the procedure below rather than the *To manually route a track* procedure given in the *Using manual routing tools* section.

Manually routing nets connected to planes and copper pours

Follow the procedure below to manually route a net whose connectivity is partially satisfied by planes and copper pours.

Note: Do not use the Minimize Connections command while routing. Running the Minimize Connections command while you are routing causes Layout to discard the copper pour connectivity database and display connections using ratsnests. If that happens, repeat steps 1 and 2.

Note: Yellow triangles in the ratsnest indicate unrouted, zero-length connections (connections that lead directly from a pad on the top layer to a pad on the bottom layer). These connections need to be routed using a via.

To manually route nets that are connected to planes and copper pours

1 Set User Preference options.

   a. From the Options menu, choose User Preferences.

   The User Preferences dialog box appears.

   b. Select the Enable Copper Pour check box.

   c. Select the Use Pours for Connectivity check box.

   d. Click OK.

2 Update the connectivity database.

   a. Choose the Refresh All toolbar button.

   Layout updates the connectivity database. Ratsnets disappear from connections that have been completed through planes and copper pours.

3 Adjust User Preference options

   a. From the Options menu, choose User Preferences.

   b. Clear the Enable Copper Pour check box.

   This ensures that copper pour won't obscure items you might want to work with, but still allow the Use Pours for Connectivity option to function.

4 Set the reconnection type

   a. Choose the spreadsheet toolbar button and choose Nets.

   The Nets spreadsheet appears.

   b. Select all nets.

   c. From the pop-up menu, choose Properties.

   The Edit Net dialog box appears.

   d. Choose the Net Reconn button.

   The Reconnection Type dialog box appears.

   e. Select the No Dyn. Reconn option.

f. Click OK in both dialog boxes.

5 Manually route the nets.

While routing, if you press the ALT key and click the left mouse button on a track, you can begin a new track. This is known as T-routing. You can use T-routing, even though dynamic reconnection is disabled.

Related topics

User Preferences dialog box

Route Settings dialog box

Using edit segment mode

Select Tool command (Track)

Select Tool command (Track Segment)

Enabling power and ground for routing

To enable power and ground for routing

1 Choose the spreadsheet toolbar button, then choose Nets. The Nets spreadsheet appears.

2 Double-click in the title cell of the Routing Enabled column. The Edit Net dialog box appears.

3 Deselect the Routing Enabled option, then choose the OK button. The Routing Enabled for all nets changes to No.

4 While the Nets spreadsheet is displayed, press the TAB key to open the Net Selection Criteria dialog box.

5 Enter VCC in the Net Name text box, then choose the OK button. The VCC net is highlighted in the Nets spreadsheet.

6 From the pop-up menu, choose Properties. The Edit Net dialog box appears.

7 Select the Routing Enabled option.

8 Choose the Net Layers button. The Layers Enabled for Routing dialog box appears.

9 Select POWER in the Plane Layers group box.

10 Choose the OK button twice to close the dialog boxes. The Routing Enabled for the VCC net changes to Yes*.

11 Repeat steps 4 through 10 for the ground net, using GND as the net name and as the plane layer.

12 Close the Nets spreadsheet.

Note: In the Nets spreadsheet, the asterisk (*) next to a Yes or No indicates that the net has special layer considerations. For example, it could indicate that the net is connected to a plane, or that one of the routing layers is disabled for the net. You can check which layers are enabled for a given net using the Enable Layers for Routing dialog box accessed through the Edit Net dialog box.

Related topics

Nets spreadsheet

Defining a DRC box

Using a DRC box, you can define the location at which you want to begin routing. The autorouter and the interactive routing tools (auto path route mode and shove track mode) run only in a DRC box. Once you start the autorouter (available in Layout and Layout Plus) it automatically begins routing the board at the area you designate. If you are manually routing, Layout zooms in to the area encompassed by the DRC box and centers it on the screen.

To define a DRC box

1. If the current DRC box is not appearing, choose the online DRC toolbar button, then choose the refresh all toolbar button. The current DRC box appears.
2. From the View menu, choose Zoom DRC/Route Box. The cursor changes to a “Z.”
3. Click the left mouse button at one corner of the box you would like to define, and while holding down the left mouse button, drag the cursor to the opposite corner of the area you would like to define, then release the left mouse button. Layout zooms in on the area, centering it on the screen.

To move a DRC box

1. If the current DRC box is not appearing, choose the online DRC toolbar button, then choose the refresh all toolbar button. The current DRC box appears.
2. From the View menu, choose Zoom DRC/Route Box. The cursor changes to a “Z.”
3. Move the cursor to the target location and click the left mouse button. Layout zooms in at the new location, centering it on the screen.

Note: To move the DRC box without zooming in, choose Zoom DRC/Route Box from the View menu, position the DRC box cursor (“Z”) over what is to be the center of the new box, type an asterisk (*) using the numeric keypad, then choose End Command from the pop-up menu.

To resize the DRC Box

1. From the View menu, choose DRC Box. The cursor shape changes to the zoom cursor.
2. Place the pointer at one corner of the new DRC Box.
3. Click and drag the pointer to resize the DRC Box.
4. Release the button when you are satisfied with the size.

Layout zooms in on that area.

Fanout

Fanout is the process of routing a surface mount device (SMD) pad to a via so that the pad can be routed on other layers. For power and ground pads, the fanout is attached to a power or ground plane using a thermal relief.

Full board fanout offers a higher probability for the router to complete signal routing for dense, multilayer SMD designs. Unlike power and ground routing, it is not absolutely necessary to implement fanout for all pads, because the router can usually successfully route those pads for which it could not place a fanout via.

Fanout is especially useful for:

- Multiple-layer boards that include power and ground planes.
- Densely packed boards that prohibit routing on the surface layers.
- Boards that include fine-pitch components that impede surface routing.
- Boards that need minimum route exposure.

Note: In previous versions of Layout, fanout capabilities were referred to as "via dispersion." In addition to the change in terminology, the implementation procedure has changed as described in the online help.

When you set fanout parameters, you can specify that Layout immediately continue with a batch route after the fanout implementation, or you can defer routing until later.

For fine pitch components, it is quite helpful to run component fanout, since this is typically the only way you can disperse all pins without blocking off one or more of the pins in the process.

You should match your via grid to your component pitch for the best fanout results.

Note: If you select a component that has been fanned out using free vias with a single mouse click, the free via fanout is included in the selection. However, if you select the component by using either area-select or shift+CLICK, Layout asks if you want to include associated free vias and test points in the selection. Choosing to exclude free vias makes editing a component using the Edit Component dialog box easier, because otherwise you're given all the properties for the free vias in addition to all of the properties for the component.

Fanout and surface mount devices

To automatically fan out surface mount devices

1. From the Options menu, choose Fanout Settings. The Fanout Settings dialog box appears.
2. Select the appropriate options, then choose the OK button.
3. From the Auto menu, choose Fanout, then choose Board.

To manually fan out surface mount devices

1. Choose a routing tool.
2. Select the VCC or GND net.
3. Route the net to the point at which you want to insert a via.
4. Press the SPACEBAR to insert a vertex (a corner).
5. From the pop-up menu, choose Add Via.
   or
   From the pop-up menu, choose Add Free Via.

Related topics

Fanout Settings dialog box

Board command (Fanout)

Component command (Fanout)

DRC/Route Box command (Fanout)

Implementing power and ground fanout

Thermal reliefs

Edit Net dialog box

Autoroute Flyout Commands

Fanning out BGAs and micro-BGAs

When fanning out BGA and micro-BGA pads, a smaller via than the one set as the default is generally required. Layout addresses this situation by letting you define the required via size and override the default via for as many components as necessary.

You should always fanout BGAs, micro_BGAs, and fine-pitch components before fanning out other components in the design.

To fanout BGAs and micro-BGAs

1 Determine the pin-to-pin spacing for all BGAs and micro-BGAs you are going to fanout.

Enable the Allow via under pad option.

1 From the Footprint spreadsheet, double-click a BGA or other footprint. The Edit Pad dialog box appears.

2 Select the Allow via under pad check box.

Ensure the required via size is defined.

1 Ensure the required via size (based on BGA specifications and DRC clearance requirements) is defined in the Padstack spreadsheet.

If the via size is not defined, follow the procedures in Defining vias.

Set the Via grid value.

1 From the Options menu, choose System Settings.

2 Set the via grid value to one-tenth the value of the pin-to-pin spacing.

Set appropriate fanout settings.

1 From the Options menu, choose Fanout Settings.

2 Select the Lock after fanout check box.

3 Enter a value into the Maximum fanout distance box:

If you want to…	do this…
fanout a BGA using the "dogbone" technology	enter the BGA pin-to-pin distance value
fanout a micro-BGA, using "via in pad" technology	enter zero as the value

4 From the Default via list, select the appropriate via.

5 Select the Override via per net check box and click OK.

Fanout the component.

1 Select the component you want to fanout.

2 From the Auto menu, point to Fanout and select Component.

Refresh the screen.

1 From the toolbar, select the Refresh All button.

You can continue selecting and fanning out as many components as you need. The settings stay in effect until you change them. If you want to fanout BGAs with different pin-to-pin spacing, ensure you change all the appropriate settings according to the procedure above.

You can also use the DRC/Route box to include multiple components.

After fanning out the BGAs and micro-BGAs, you can set the via grid to other appropriate values to complete fanning out the board.

Related topics

Edit Net dialog box

Fanout Settings dialog box

Footprints spreadsheet

Board command (Fanout)

DRC/Route Box command (Fanout)

Implementing power and ground fanout

Thermal reliefs

Implementing power and ground fanout

You must implement power and ground fanout before you route any signals. Power and ground fanout connects all surface mount power and ground pins to the appropriate internal plane layer using a via for thermal relief connection.

Note: In previous versions of Layout, fanout capabilities were referred to as "via dispersion." In addition to the change in terminology, the implementation procedure has changed as described in the online help.

To perform power and ground fanout

1. Using the Nets spreadsheet (SHIFT+N), edit all power and ground nets to attach them to the appropriate plane. This makes it possible for the router to recognize these nets as voltage nets rather than signal nets.
2. Disable routing for all nets except the power and ground nets. You can select all nets except power and ground by clicking the left mouse button in the Net Name column, then pressing SHIFT and clicking on the power and ground nets to deselect them.
3. From the Auto menu, select the appropriate fanout command (board, DRC/Route Box or component). Layout displays the Fanout SMD Pads dialog box.
4. Select the Fanout Power/Gnd option and all the appropriate options within that group box, and deselect the Fanout Signals option.
5. Set the Fanout Direction options as desired.
6. In the Maximum Fanout Distance text box, specify the maximum distance you want to allow between the SMD and the fanout via.
7. Select the Start batch route when done option to automatically run the autorouter after completing the fanout and choose the OK button.

   or

   If you want to delay autorouting, choose the OK button without selecting the Start batch route when done option.

   Layout implements the power and ground fanout according to the options you set.
8. Manually or interactively route all failures. Use Bad SMD-to-Plane connection error markers to locate the failures.
9. Run the Board Design Check with SMT Fanouts selected. This determines if all power and ground pins are dispersed to a plane. Repeat steps 8 and 9 until all pads are properly attached.

Note: You can tile the Error Markers spreadsheet and the design window. Then, when you double-click on an error, the design window will zoom to the appropriate pad.

Related topics

Fanout

Edit Net dialog box

Load Strategy command (File menu)

Board command (Autoroute)

Thermal reliefs

Fanout Settings dialog box

Board command (Fanout)

Component command (Fanout)

DRC/Route Box command (Fanout)

Routing

You can use the Select Tool command (Track) command to create new tracks, or edit existing tracks without ripping them up, by selecting any routed vertex or segment and clicking the left mouse button. If you select a track using Manual Route, you can continue routing the net one segment at a time, at a 45-degree or 90-degree angle.

To manually route

1. From the Tool menu, point to Track and choose Select Tool.
2. From the View menu, choose Zoom In, click on the screen to magnify the routing area you will be working in, then press ESC to end the zoom command.

3 Select a ratsnest. The cursor changes to a small-size cross and the ratsnest is attached to the pointer.

4 Drag the pointer to draw a track on your board. When you click the left mouse button, the track is anchored at the nearest pad on the net.

5 Continue to move the cursor to draw additional segments of the track, clicking the left mouse button to create vertices (corners) in the track as you route. You can delete a routed segment by placing the cursor over the segment and pressing DELETE.

   Layout can provide directional hints (Routing hints) for exiting pads, vias, and routed segments. You can enable routing hints in the Route Settings dialog box.

6 Click the left mouse button to complete the route. The cursor changes to a regular-size cross and the ratsnest disappears from the cursor.

7 Select the completed route and choose Lock from the pop-up menu. This locks the route you created so that it cannot be moved if the board is later autorouted.

Note: Yellow triangles in the ratsnest indicate unrouted, zero-length connections (connections that lead directly from a pad on the top layer to a pad on the bottom layer). These connections need to be routed using a via.

When modifying tracks, you may see slightly different effects, depending on whether you are editing a segment or a vertex. For example, if you place your pointer directly on top of a vertex and click the left mouse button, the router picks up only the vertex and the two immediately adjacent segments. You can specify the effect of route selection in the User Preferences dialog box.

When you select a track with the left mouse button at a location where there is copper on more than one layer, the router edits the track that is on the current layer.

If you pick up an existing track and type a layer number, the track switches to the new layer, and vias are installed automatically where necessary. If it is impossible to clear room

for the vias, the router responds with beeps and does not switch the track.

You can copy tracks by selecting an area (even if that area encompasses only one track), and pressing INSERT. A ratsnest connection between two pads must already exist in the location to which you copy the track. Copying tracks is particularly useful for any repeated circuitry, such as round IC test boards with repeated circuitry.

To create a track with curved corners

1. From the Tool menu, point to Track and choose Select Tool, or from the toolbar, select the Add/Edit Route Mode button.
2. Click the right mouse button, and from the pop-up menu, choose Curved Corners.
3. Follow the procedures for routing point-to-point connections above. Vertex corners will be curved.

To change the track width while you are routing

1. After placing a vertex before the position where you need to change the track width, click the right mouse button, and from the pop-up menu, choose Change Width. The Track Width dialog box appears.
2. Enter an integer value in the New Width text box.
3. Click OK.

   Layout uses the new width for all subsequent segments, or until you change the width again.

To change the layer of a routed net

1. Verify that one of the routing tools is active.
2. Control-click to select the net.
3. Press the layer number shortcut key for the new layer.

   or

Choose the new layer from the Layer drop-box on the toolbar.

4 The net switches to the new layer, and vias are installed automatically where necessary. If it is impossible to clear room for the vias, Layout beeps and does not switch the net.

To change the track width of a routed net

1 Verify that one of the routing tools is active.

2 Select the net.

3 Click the right mouse button, and from the pop-up menu, choose Change Width. The Track Width dialog box appears.

4 Enter an integer value in the New Width text box.

5 Click OK.

The width shown in the Con Width field in the Nets Spreadsheet is the default width for any new segments. If you wish to change the default, so that you do not have to display the Track Width dialog box each time you begin routing a net, you should change the Con field for the appropriate net.

Related topics

Place Settings command

Route Settings dialog box

Using edit segment mode

Select Tool command (Track)

Select Tool command (Track Segment)

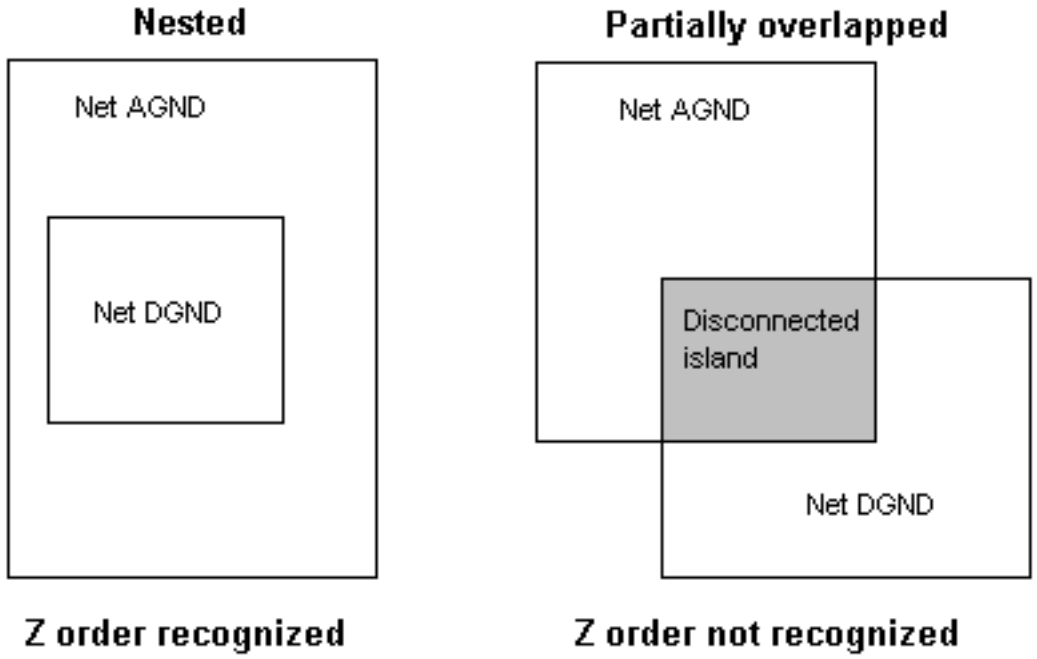
Routing to an off-grid pad

Creating split planes

By placing copper pours on a plane, you can create as many isolated areas as necessary to accommodate power and ground requirements, and connect specific nets to them. To connect the specific nets to the copper pours, you first assign a primary net to the plane layer by using the Edit Net dialog box, and then connect secondary nets by indicating the net name in the Edit Obstacle dialog box and including at least one pin of the net when drawing the copper pour.

You can also create nested copper pours on a plane and assign them a Z order. Layout nests a higher-numbered Z order pour within the next lower-numbered Z order pour, and applies the applicable clearance rules.

Note: On a plane layer, Layout recognizes Z order with nested copper pours, but not with pours that partially overlap. Incomplete overlapping causes a disconnect-island between the overlapping pours.



To create split planes

1 Select the primary net for attachment to the plane layer.

   a. From the toolbar, select the View Spreadsheets button.

   b. Choose Nets.

   c. Select the primary net for assignment to the plane layer.

   d. Click the right mouse button and from the pop-up menu, choose Properties.

   The Edit Net dialog box appears.

2 Attach the primary net to a plane layer.

   a. Click the Net Layers button.

   The Layers Enabled for Routing dialog box appears.

   b. In the Plane Layers frame, select the appropriate plane layer (PWR or GND) for the primary net.

   You can change routing layers or spare layers into plane layers using the Layers spreadsheet. For information on designating layers as plane layers, see the Edit Layer dialog box topic.

   c. Click OK in both dialog boxes.

   This causes Layout to place thermal reliefs on the plane layer for the through-hole pads and vias of the primary net.

3 Assign distinctive colors to the nets.

   a. While still in the Net spreadsheet, right click the mouse over the Color cell of the primary net or a secondary net.

   b. Choose Change Color.

   c. Select a color from the color box.

   d. Repeat steps a, b, and c for each associated net, selecting a different color for each.

The thermal reliefs for the primary net appears in the color you selected for it. Thermal reliefs for secondary nets will appear in their respective colors after you connect them to a copper pour and choose the Refresh All command (or toolbar button).

4 Create a copper pour and attach a secondary net to it.

a. From the View menu, choose Zoom DRC/Route Box.

b. Zoom in on the area to be affected by the copper pour.

Ensure at least one pin in the net you want to associate with the copper pour is showing on the screen.

c. Choose the Obstacle toolbar button.

d. From the pop-up menu, choose New.

e. From the pop-up menu, choose Properties.

The Edit Obstacle dialog box appears.

f. From the Obstacle Type drop-down list, select Copper Pour.

g. From the Net Attachment drop-down list, select the net you want to attach to the copper pour.

h. Draw an outline for the copper pour, including at least one pin of the attached secondary net.

5 Refresh the copper pour and connectivity.

a. From the Auto menu, point to Refresh and select All.

Layout updates connectivity and displays all thermal reliefs in their respective colors.

Note: In the User Preferences dialog box, you must keep the Enable Copper Pour option selected, but deselect the Use Fast Fill Mode option before you create a Gerber plot. Otherwise, your Gerber plots will have no copper pours.

Related topics

Edit Layer dialog box

Board command (Fanout)

DRC/Route Box command (Fanout)

Layers spreadsheet

Layers Enabled for Routing dialog box

Thermal Relief Settings command

Gerber plot preview

Creating a copper pour

Verifying plane layer connections and disabling power and ground nets

To verify connections to the planes

1. Choose the spreadsheet toolbar button, then choose Statistics. The Statistics spreadsheet appears.
2. If necessary, respond to the message asking if you want to repour copper by choosing the Yes button.
3. Scroll until you find the Routed row, which is the beginning of the routing data. You should see a value of 100% in the Enabled column for % Routed, which indicates that the appropriate nets are connected to the plane layers.
4. If the value is anything less than 100%, choose the refresh all toolbar button.
5. If the value is still anything less than 100%, minimize the Statistics spreadsheet, choose a routing tool, and route the net to the appropriate plane layer.
6. Maximize the Statistics spreadsheet, then choose the refresh all toolbar button.
7. After you've verified that the value in the Enable column for % Routed is 100%, close the Statistics spreadsheet.

To disable the power and ground nets and enable other nets

1. Choose the spreadsheet toolbar button, then choose Nets.
2. Click once in the title cell of the Routing Enabled column. The entire column is highlighted.
3. From the pop-up menu, choose Enable<->Disable. The Routing Enabled for the VCC and GND nets changes to No*, and the Routing Enabled changes to Yes for the rest of the nets.

Using manual routing tools

You can use add/edit route mode to create new tracks from a ratsnest. To edit existing tracks without unrouting them, place your cursor on any routed vertex or segment and click the left mouse button.

You can use edit segment mode to move existing segments of tracks, create new segments, or remove segments. When a horizontal segment is moved up or down, the connecting segments lengthen or shorten in order to accommodate the changes to the selected segment. The selected segment and its connecting segments change size as necessary.

Note: While routing, if you press the ALT key and click the left mouse button on a track, you can begin a new track on another track of the same net, which is known as T-routing.

When using the manual route tools, the following options are available in the Route Settings dialog box (from the Options menu, choose Route Settings).

- The Use All Via Types option allows Layout to use the optimal via type from among all the vias defined in the Padstacks spreadsheet. If this option is not selected, and you have not specified a via for use with a given net, then Layout uses Via 1 (the default via type).
- With the Snap to Grid Routing option selected, the segment that you are routing moves from grid point to grid

point, so that you cannot create a track off of the routing grid. When you deselect this option, you are able to route regardless of the track's relationship to the routing grid.

- The Any Angle Corner option allows you to create an angle of any kind. When you select this option, the connection segment attached to the routing tool's crosshairs rotates freely through 360˚.
- The 45 Corners option allows you to create angles of 90˚ or 135˚ while you route.
- The 90 Corners option restricts angles to 90˚.
- The Curve Corners option gives you the ability to place curved tracks on your board while you route manually. With a routing tool selected, you can create curved, horizontal and vertical tracks (however, you cannot readily create 135˚ angles with this option selected).

Using add/edit route mode

You can use the add/edit route mode to route new tracks and edit existing tracks. If you select a partially routed track, you can continue routing the track, one segment at a time, at a 135˚ or 90˚ angle. When you select a track at a location where there is copper on more than one layer, the router edits the track that is on the current layer.

If you pick up an existing track, press the SPACEBAR, and type a layer number, the track switches to the new layer, and vias are installed automatically where necessary. If it is impossible to clear room for the vias, the router responds with beeps and does not switch the track.

Note: By default, DRC (Design Rule Check) is always on for routing. To disable it, choose the online DRC toolbar button. The words "DRC OFF" display in the design window's title bar.

To manually route a track

1 Choose the add/edit route toolbar button.

2 Choose the zoom in toolbar button, then click the left mouse button to magnify the area to route. Press ESC to exit zoom mode.

3 Select a ratsnest with the left mouse button. The ratsnest attaches to the pointer.

4 Drag the pointer to draw a track on the board.

5 Click the left mouse button or press the SPACEBAR to create vertices (corners) in the track.

6 When drawing the last segment for the connection, choose Finish from the pop-up menu. The track automatically connects to the center of the pad. A complete connection is indicated by the cursor changing size and the ratsnest disappearing from the pointer.

Note: You can also copy tracks, which may be useful for certain boards, such as round IC test boards with repeated circuitry. For information on copying tracks, see *Copying tracks* on page 412.

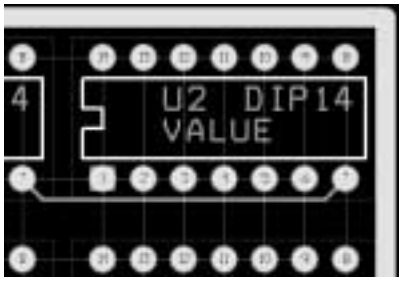
Note: The final segment must meet the target pad at a 90˚ or 135˚ angle to finish.

Using edit segment mode

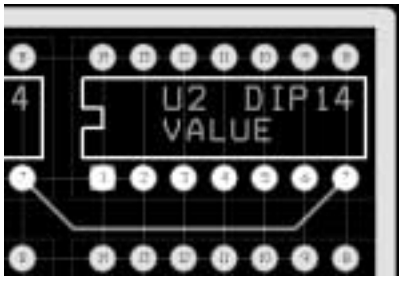
Layout track segments consists of three areas: two end areas and a center area. Selecting the center area of a segment selects the entire segment. You can reposition the segment by selecting the center area. Selecting the ends of segments lets you change segment length and direction, and vertex angles.

When moving segments, Layout creates angles based on the manual route settings, and maintains legal routing patterns. It will not normally create acute, non-orthogonal, or non-135 degree angles. In rare cases, it may create an acute angle if it is a useful interim step toward accomplishing the final routing goal.

Note: The add/edit route mode can temporarily enter edit segment mode. For information on this, see *Moving segments of tracks* on page 414.

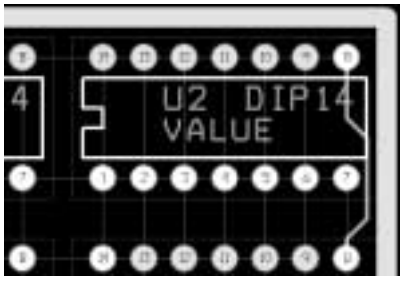


Original segment

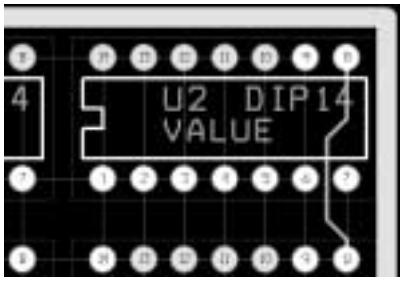


Segment after moving

Layout moves the segment only when the pointer is within “picking distance” of an acceptable area of the board for relocating the track. When you move the pointer over a legal potential track, Layout moves the selected segment to that location. Using this method, you can jump over an intervening pad or via.

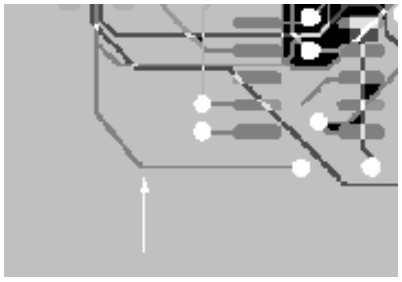


Original segment

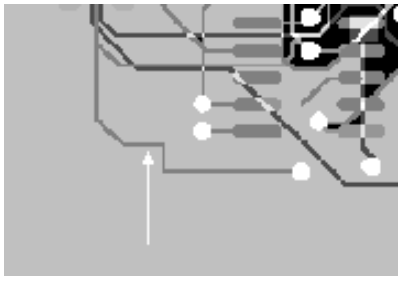


Segment after moving

Selecting an end segment adds a vertex at the point of selection.



Original segment



Segment after moving

To move a segment

1. Select in the middle of the segment you want to move.
2. Move the segment to the desired location.
3. Press the left mouse button.

To change the width of a segment you are installing

1. Select a connection for routing.
2. Click the left mouse button to accept the track up to the point where you want to change the width.
3. Press w. Layout displays a dialog box showing the present width of the line.
4. Enter the new width. It will overwrite the old width.
5. Click OK.

If you place the pointer on an unselected track segment and press w, a dialog box is displayed letting you change the width of the segment, the connection, or the entire net.

If you have enabled online DRC, which you can enable from the User Preferences dialog box, Layout displays a warning if you are about to create a short.

Related topics

Using edit segment mode

Routing

Routing hints

Route Settings dialog box

Using interactive routing tools

Online DRC (design rule check) is automatically activated whenever you choose either of the interactive routing tools

(shove track or auto path route). In addition, you can only use the interactive routing tools on connections within the DRC box.

Note: For information on the DRC box, see *To define a DRC box*.

Shove track mode is considered interactive routing because you are interacting with the automatic push-and-shove routing capabilities of Layout when you are routing a track.

Auto path route mode (not available in Layout Engineer's Edition) is considered interactive routing because you are interacting with the autorouter when it suggests tracks and suggests via placement (if you select the Suggest Vias option in the Route Settings dialog box.

Using shove track mode

When you use shove track mode, Layout shoves other tracks out of the way of the track that you are currently routing. With this mode, you can pick up individual connections and route them aided by the shove capability, manually route critical tracks, and edit tracks and vertices.

To set routing parameters for shove track mode

1. From the Options menu, choose Route Settings. The Route Settings dialog box appears.
2. Select the Shove Track Mode option, select one of the following options, then choose the OK button.

Low Power	The router moves tracks only slightly, or conservatively, in an attempt to move them out of the way as you add new tracks.

Medium Power	The router shoves tracks, and may even push routes over other items (such as pads) and around other tracks in an attempt to move them out of the way as you add new tracks.
High Power	The router rips up, shoves, and reroutes existing tracks as you add new tracks.

To use shove track mode

1. Choose the shove track toolbar button.
2. Define the DRC box size to encompass your area of interest.
3. Select a connection with the left mouse button. The connection attaches to the pointer.
4. Drag the pointer to draw a track on the board.
5. Click the left mouse button or press the SPACEBAR to create vertices (corners) in the track.
6. When drawing the last segment for the connection, choose Finish from the pop-up menu. The track automatically connects to the center of the pad. A complete connection is indicated by the cursor changing size and the ratsnest disappearing from the pointer.

Note: When you use shove track mode, the router does not automatically show you where vias are needed. To change layers while routing a track, press the key corresponding to the target layer (for example, to change to the bottom layer, press 2). The router clears away tracks around the via you are inserting when you click the left mouse button to accept the first segment on the new layer.

Using auto path route mode

When you use auto path route mode (not available in Layout Engineer's Edition), Layout suggests a possible track when you select a ratsnest or pin. As you move the cursor, the suggested track changes position. When you click the left mouse button, auto path route mode places the suggested track using the push-and-shove routing capabilities of the autorouter, thereby clearing away any imposing tracks. Note that the final track may not look like the suggested track. You can only use auto path route mode with online DRC enabled. Attempting to disable online DRC takes you out of auto path route mode.

Note: If you double-click on a connection, auto path route mode routes the track for you automatically.

When you use auto path route mode with the Suggest Vias option selected in the Route Settings dialog box (from the Options menu, choose Route Settings), Layout displays potential via locations as you're routing, and removes them if they're not needed in the final version of the track.

To set interactive autorouting options for auto path route mode

1. From the Options menu, choose Route Settings. The Route Settings dialog box appears.
2. Select the Auto Path Route Mode option, select one of the following options, then choose the OK button.

Allow Off-Grid Routing	This option allows auto path route mode to display possible routing paths without regard to the routing grid. Selecting this option is the only way to permit auto path route mode to end tracks at an obscure angle of approach. Off-grid routing is almost always needed for mixed-pitch boards.

Shove Components	This option allows auto path route mode to shove components in much the same way as it shoves tracks. That is, when you place a vertex using the left mouse button or SPACEBAR, any imposing components are moved away from the vertex (unless those components are locked).
Maximize 45 Corners	This option allows auto path route mode to optimize routing space with vertices of 135˚ or 90˚. If deselected, the autorouter creates 90˚ corners only.

Creating duplicate connections

You have the ability to insert a *duplicate connection* from a pad, a vertex, or a corner. A duplicate connection is a redundant circuit, or two tracks that connect to the same pads at both ends. Using this ability, you can insert guard ring connections for shielding, meet special routing requirements, or split nets.

To create duplicate connections, you first route the ratsnest between the two pads. Then, you use the connection toolbar button to place a second ratsnest between the two pads, and then route the second ratsnest.

To create a duplicate connection

1 Choose the zoom in toolbar button, then area-select the target pads, to magnify them on the screen.

Note: If the reconnection type for the net is set to something other than None or High speed, you can use Layout's dynamic reconnect to shorten the procedure above. Route the first connection between the two pads. Then, pick up a connection leaving one of your two pads (you may have to choose Exchange Ends from the pop-up menu to get the "outgoing" leg). Route it to the

target pad. Once the end of the track gets closer to your new target pad than it is to the original pad, the connection jumps to the pad for which you want the duplicate connection.

2 Choose one of the routing toolbar buttons.

3 Create a track between the two pads by routing the existing connection.

4 From the pop-up menu, choose Lock.

5 Choose the connection toolbar button.

6 Create a new connection between the two pads.

7 From the pop-up menu, choose End Command.

8 Choose one of the routing toolbar buttons.

9 Create a track between the two pads using the connection you added.

10 From the pop-up menu, choose Lock.

Manual routing techniques

There are several commands available on the Edit menu and pop-up menus to assist you in routing a board. Use the following commands and techniques to route your board.

Minimizing connections

The Minimize Connections command finds the shortest connection possible for each connection in the ratsnest. If you have nothing selected, it reconnects the entire board. If you have a net selected, it will minimize the connection for just that net.

To minimize connections

1 From the pop-up menu, choose Minimize Connections.

Changing the colors of nets

To change the color of a net

1. Choose the spreadsheet toolbar button, then choose Nets. The Nets spreadsheet appears.
2. Select a net in the spreadsheet, then choose Change Color from the pop-up menu.
3. Select a color from the color palette that appears. The net changes to the new color.

Copying tracks

You can copy multiple tracks, which is how you route duplicate channels of circuitry. A ratsnest connection between two pads must already exist in the location to which you copy the track. Copying tracks is particularly useful for any repeated circuitry, such as round IC test boards with repeated circuitry.

To copy multiple tracks

1. Choose one of the routing toolbar buttons.
2. Area-select one or more tracks.
3. Press CTRL+C or the INSERT key to copy the tracks. The tracks attach to the pointer.
4. Press the left mouse button to paste the track. Note that the arrangement of the target pads must match the arrangement of the source pads for the paste to complete.
5. From the pop-up menu, choose End Command.

Removing tracks

There are some options available for “undoing” the routing performed on a track if you are not achieving the desired results. With one of the manual route tools active, commands

for unrouting segments or tracks are available on the pop-up menu.

- Unroute Segment rips up the segment “behind” the one you are dragging (the segments drawn before the current segment), and continues to rip up segments back to their source if you continue to use Unroute Segment. If you are using the DRC-enabled environment, the ripup stops at the DRC box edge.
- Unroute rips up the track for the entire connection. If you are using the DRC-enabled environment, the ripup stops at the DRC box edge.
- Unroute Net rips up the tracks for the *entire* net, regardless of whether you are in the DRC-enabled environment or not.

To unroute routed segments or tracks

1. Select a track.
2. From the pop-up menu, choose Unroute Segment, Unroute, or Unroute Net.

There are also commands for removing whole and partial routes that you can access from the pop-up menu when the Nets spreadsheet is open.

- Unroute Partial Track removes routes that are not complete.
- Unroute Center Partial removes routes that are not connected to a pad at either end.
- Unroute removes the routes for the entire net.
- Unroute Unlocked Track removes unlocked routes from the board.

To unroute routed tracks in the Nets spreadsheet

1. Open the Nets spreadsheet.

2 Select one or more nets. If you want the command to affect the entire board, click once in the Net Name title cell.

3 From the pop-up menu, choose Unroute Partial Track, Unroute Center Partial, Unroute, or Unroute Unlocked Track. The routed segment or entire route of the track is removed, but the net remains on the board and in the Nets spreadsheet.

Note: To unroute the entire board, from the Auto menu, choose Unroute, then choose Board. To unroute all copper on a net, place the pointer somewhere over the net but don't select it, then press D.

Related topics

Nets spreadsheet

Moving segments of tracks

Choosing the edit segment toolbar button puts you in edit segment mode, which you should use to move existing tracks. However, if you choose the add/edit route toolbar button to enter add/edit route mode, then select a segment, you can temporarily enter edit segment mode by choosing Segment from the pop-up menu. You remain in edit segment mode only until you click the left mouse button, at which time you are returned to add/edit route mode.

Note: If you are in edit segment mode and choose a connection instead of a track segment, you are put into add/edit route mode for the current connection only. You can use this to your advantage if you are editing a segment and can't get it in the position you want. While in edit segment mode, double-click on a segment to enter add/edit route mode, then route the connection the way you want it.

To move a segment

1 Choose the edit segment toolbar button.

2 Select a segment and slide it as desired

Or

3 Choose the add/edit route toolbar button.

4 Select a segment.

5 From the pop-up menu, choose Segment.

6 Slide the segment as desired.

Changing the widths of tracks

The Change Width command changes the width of the segment you are currently routing. This command temporarily overrides any value you may have set in the Net Widths By Layer dialog box (accessed by double-clicking in a cell in the Nets spreadsheet, then choosing the Width By Layer button).

To change the width of a track

1 Select a track.

2 From the pop-up menu, choose Change Width. The Track Width dialog box appears.

3 Enter a new width for the track and choose the OK button.

Related topics

Nets spreadsheet

Forcing a net width on a layer

When you set your net properties before routing, you may have specified a width for a particular net on a given layer. If you interactively change the width of the net using the Track Width dialog box, you can use the Force Width by Layer command to force a specified net width on a given layer.

To force a net width on a layer

1 Open the Nets spreadsheet.

2 Select the net with the new width in the spreadsheet.

3 From the pop-up menu, choose Force Width by Layer.

Related topics

Nets spreadsheet

Drawing arcs

Use Curved Corners command from the manual routing tools pop-up menus to add an arc to segments.

To add an arc to an existing segment

1 From the toolbar, select the Add/Edit Route Mode button.

2 Select the track segment where you want to add curved corners (arcs).

3 From the pop-up menu, choose Curved Corners.

4 Move the segment endpoints to see the curved corners.

5 Click the left mouse button again to release the arc.

Related topics

Using edit segment mode

Adding vias

The Add Via and Add Free Via commands insert a via or a free via at the last vertex you created. This is useful for manually creating dispersion vias, which are short connections from SMDs to the power and ground planes.

To add a via

1. Select a track.
2. Insert a vertex by clicking the left mouse button or pressing the SPACEBAR.
3. Type the number of the target layer to change to (the layer numbers are available on the layer drop-down list on the toolbar).
4. From the pop-up menu, choose Add Via or from the pop-up menu, choose Add Free Via.
   Layout adds a via. In the case of free vias, Layout adds a via marked with the letters “FV.”

Adding a free via matrix

Sometimes, you may want to “stitch together” plane layers with free vias. Or, you may want to add free vias around the perimeter of a copper area between multiple layers, like a Faraday cage. Layout allows you to add a free via matrix within an area-selection box or in a copper area obstacle.

A free via matrix allows you to define an area in which you want to place free vias using spacing you supply. Note that Layout only places a via where it can do so without creating a DRC violation.

Within an area-selection box, the matrix uses the net you specify in the Free Via Matrix Settings dialog box. In a copper area obstacle, however, the matrix uses the net of the obstacle, overriding any specification you made in the dialog box.

A free via matrix that is connected only by unrouted connections is never removed by AutoECO, unless the entire net is removed from the board.

To add a free via matrix

1. From the Options menu, choose Free Via Matrix Settings.
   The Free Via Matrix Settings dialog box appears.

2 Modify the settings (choose the dialog box's Help button for an explanation of the dialog box's options), then choose the OK button.

3 From the Auto menu, choose Place, then Free Via Matrix.

4 Draw an area-selection box or select a copper area obstacle.
Layout places a matrix of free vias (marked with the letters "FV") within the area-selection box or the copper area obstacle. Depending on whether you have the Periphery Only option selected in the dialog box, the matrix either fills the area or rings the periphery.

Changing vias

The Change Via Type command displays the Via Selection dialog box, within which you can select a new via type. The dialog box only displays vias that have been defined in the Padstacks spreadsheet, and are therefore available for routing.

Note: For information on defining vias, see Defining vias.

To change a via

1 Select a via by clicking on the intersection of the segments with the left mouse button.

2 From the pop-up menu, choose Change Via Type. The Via Selection dialog box appears, listing all of the vias that are available for routing.

3 Select a new via and choose the OK button.

Changing free vias

The Properties command displays the Edit Free Via dialog box, in which you can select a new free via type or edit properties of the existing free via. The dialog box only displays vias that have been defined in the Padstacks spreadsheet, and are therefore available for routing.

Note: For information on defining free vias, see *To define a free via*.

To edit a free via

1. Select a free via by clicking on the intersection of the segments with the left mouse button.
2. From the pop-up menu, choose Properties. The Edit Free Via dialog box appears.
3. Edit the following options, then choose the OK button.

Padstack Name - Select the name of a padstack type from the drop-down list. Free vias can only be assigned padstack types that are defined in the Padstacks spreadsheet.

Net Name - Free vias must be assigned to a net, regardless of their connectivity. Use the drop-down list to designate an associated net for the free via.

Convert to Component - Choosing this button displays the Select Footprint dialog box. After selecting a library, choose a footprint for the free via, then choose the OK button.

Group Number - It's possible to associate a free via with a component group while working in Layout (though it is recommended that you create groups at the schematic level). Enter the group number you want assigned to the applicable free via.

Location - The text boxes allow you to designate the X and the Y coordinates for the repositioning of a free via. If you leave these boxes blank and choose the OK button, the free via you modified moves with your pointer until you place it on your board by clicking the left mouse button.

Locked - This option locks the relevant free via in position after you place it on a board.

Using tack points

The Tack command allows you to “tack” ratsnest lines out of the way. Use this command when you need to select something under a connection.

To use a tack connection

1. Select a ratsnest line.
2. From the pop-up menu, choose Tack.
3. Drag the ratsnest line out of the way and click the left mouse button to place it. The ratsnest line is “tacked” out of your way.

To remove a tack connection

1. Select a tacked ratsnest line.
2. Choose the spreadsheet toolbar button, then choose Nets. The Nets spreadsheet appears with the selected net highlighted in the spreadsheet.
3. From the pop-up menu, choose Remove Tack Point. The last tack you added to the connection is removed.

Note: You can also remove all of the tack points on the board at once. Without selecting any nets on the board, choose Remove Tack Point from the Nets spreadsheet pop-up menu.

Related topics

Nets spreadsheet

Exchanging the ends of a connection

The Exchange Ends command exchanges the source and target of the connection so that you can route in the opposite direction. For example, if you are routing a connection and you accidentally pick up the wrong end, you can use this command to swap ends without releasing the connection.

To exchange the ends of a connection

1 Select a ratsnest line.

2 From the pop-up menu, choose Exchange Ends.

Note: When you are routing a track, if the router is not showing you exactly the path you would like, use the Exchange Ends command. This gives you two distinct sets of paths to choose from.

Locking routed tracks

The Lock command locks the selected segment, and everything behind it, back to the source point.

To lock routes

1 Select a track.

2 From the pop-up menu, choose Lock.

To unlock routes

1 Select a track.

2 From the pop-up menu, choose Unlock.

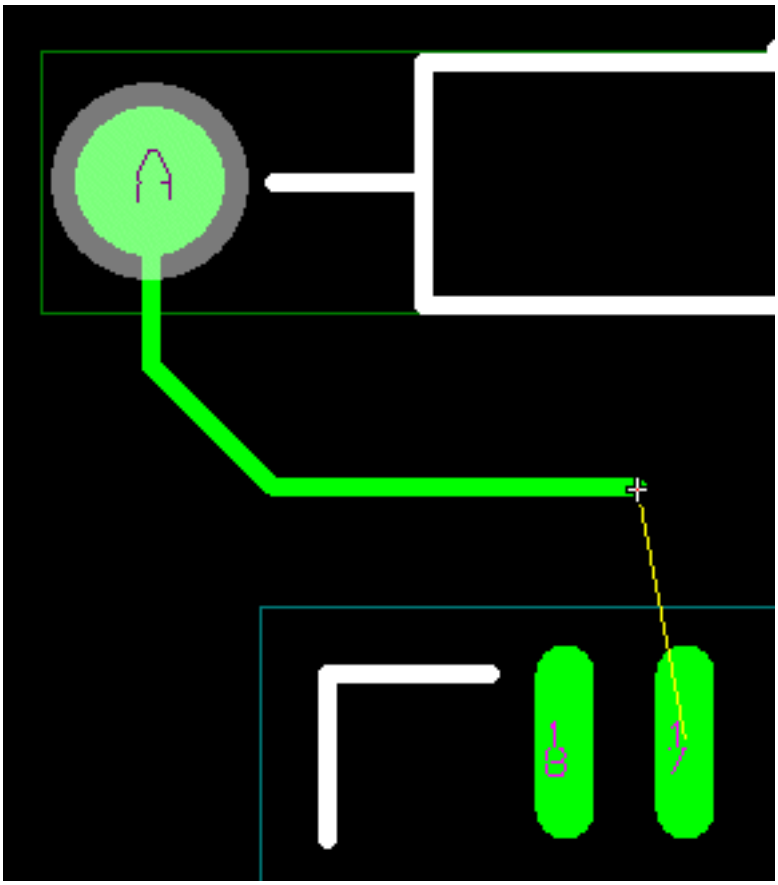
Routing to an off-grid pad

When manual routing, there are times when you may need to route to a pad that isn't set directly on the grid. Creating 45-degree angle routes in these situations can be difficult. The following procedure allows you to create clean routes to an off-grid pad.

To route to an off-grid pad

1 From the toolbar, click the Add/Edit Route button.

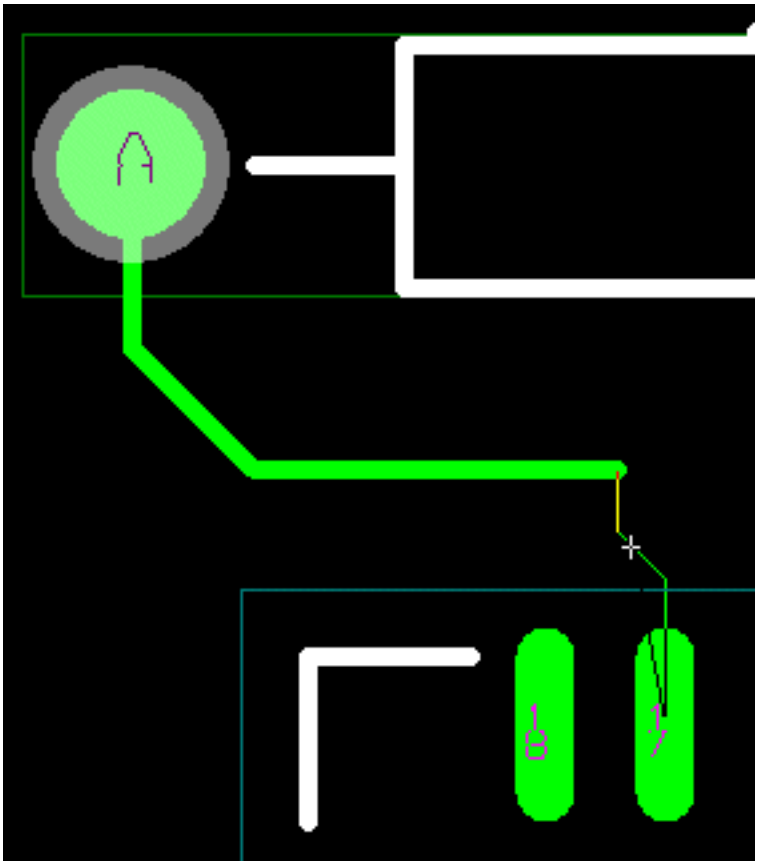
2 Click the on-grid end of the ratsnest and route toward the off-grid pad.



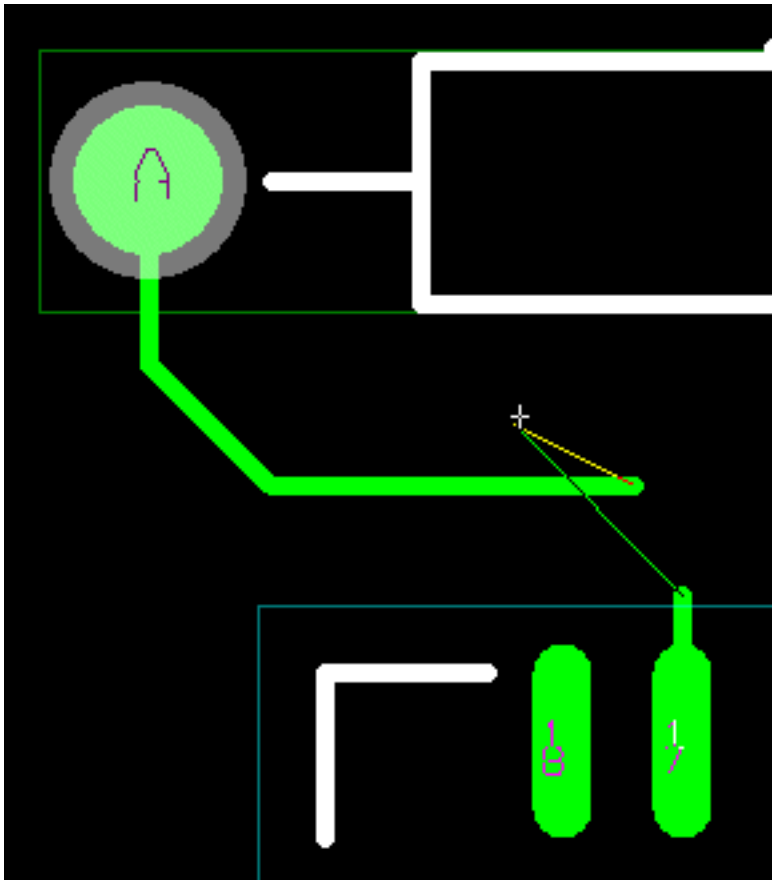
3 On the keyboard, press the X key to begin routing from the other end of the ratsnest.

or

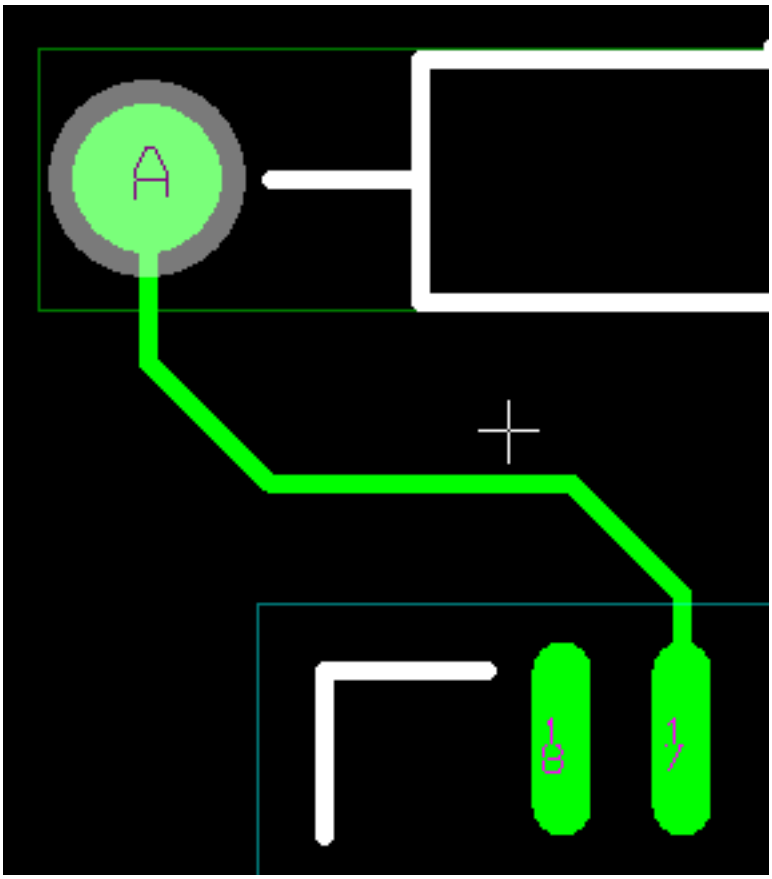
Choose Exchange Ends from the pop-up menu.



4 Click to route from the other end of the ratsnest. Place the route so that it crosses the already routed portion of the net.



5 Click to complete the route. Layout trims the route and creates a track segment at the appropriate angle.



Related topics

Routing

Teeing into or out of an existing track

Although Layout shows ratsnest lines running from pad to pad or from a pad to a fanout via, you may wish to instead route to an existing track.

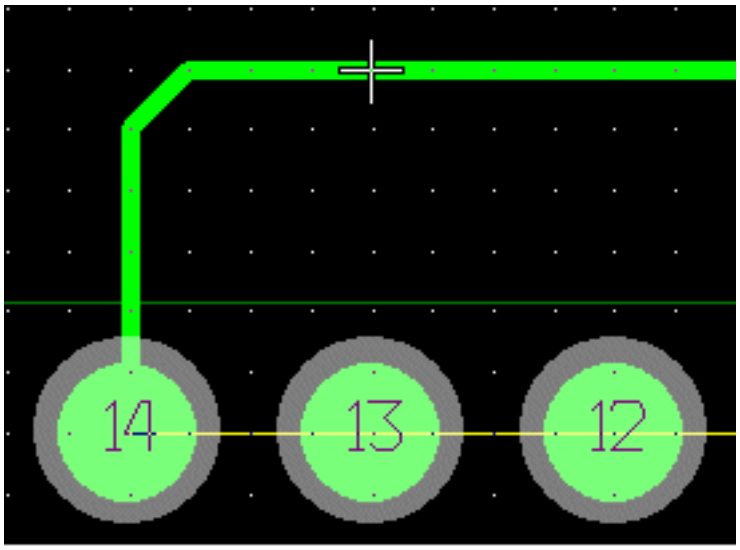
To Tee into an existing track, click on a ratsnest to route and finish the route by clicking on the interior of the track into which you want to Tee. To Tee out of an existing track, hold down ALT

and click on the portion of the track where you want to begin routing.

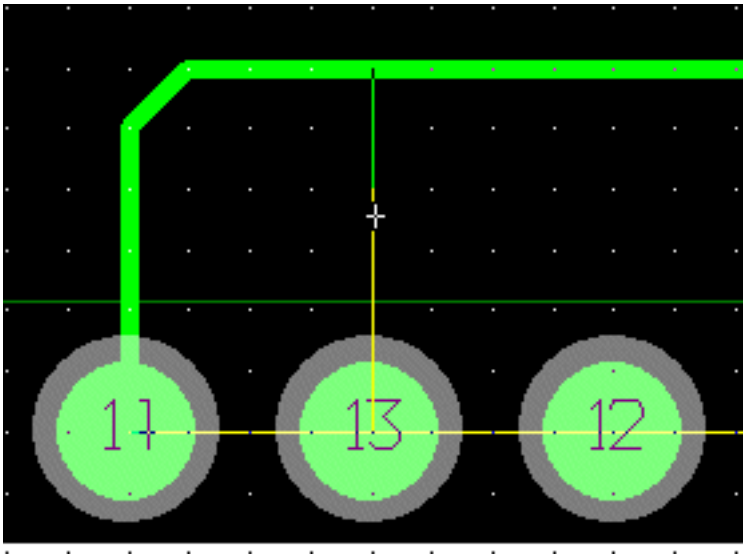
Note: Teeing is only possible if a net is not marked as "high speed". The "high speed" property can be found on the Reconnection Type dialog box.

To tee out of an existing track

1. Choose one of the routing toolbar buttons.
2. Hover over the location in the existing track where you want to connect.

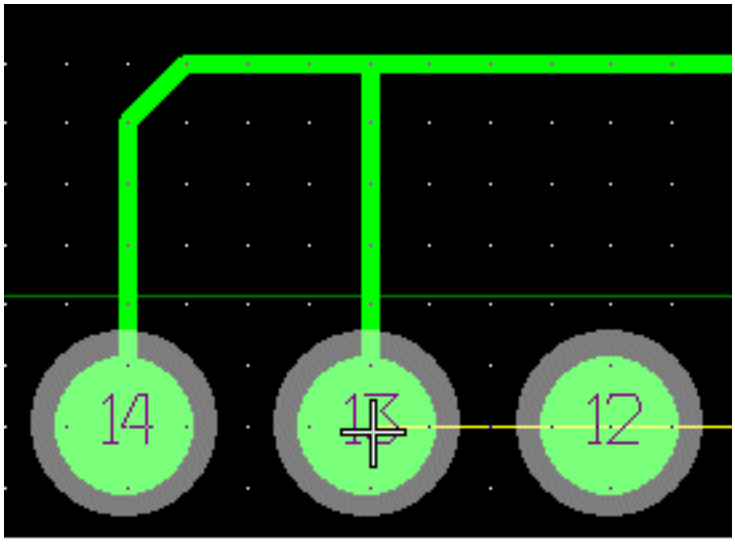


3 Hold down the ALT key and click the track.



This creates a new connection to the track. Dynamic reconnection creates a ratsnest to the nearest possible routing point, unless it has been disabled for this net in the Nets spreadsheet.

4 Route to desired pad and click to finish the route.



Related topics

Routing

Making a ratsnest invisible

If you are trying to route a particular net, and are having trouble isolating it because all of the ratsnest is visible, you can suppress the display of the ratsnest by following the steps below.

To make a ratsnest invisible

1. Choose the Spreadsheet toolbar button. The spreadsheet menu displays.
2. Choose Nets. The Nets spreadsheet displays.
3. From within the spreadsheet, select all of the nets, display the pop-up menu, then choose Enable <-> Disable. All of the nets become disabled and therefore invisible.
4. From within the spreadsheet, select the net you want to make visible, display the pop-up menu, then choose Enable <-> Disable. The net becomes enabled and therefore visible.
5. Close the Nets spreadsheet.

Related topics

Nets spreadsheet

Routing hints

Routing hints allows you to give the manual routing tool a directional "hint" for proceeding out of a pad, via, or vertex.

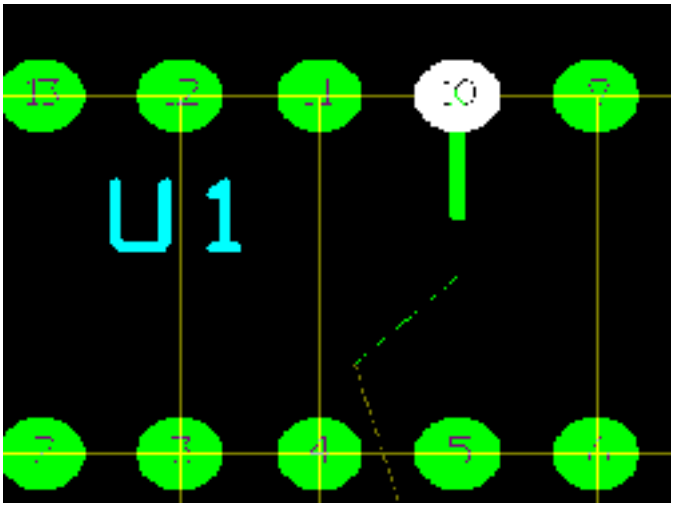
For instance, when exiting a pad, you can typically route in one of two ways. You can either exit at a diagonal and then make a 135 degree turn to go straight, or you can exit straight out and then make a 135 degree turn to a diagonal. Either way

you end at the same location. By moving the pointer in the direction you want to exit the pad and have the first segment continue, the router understands the path to take to reach that common location.

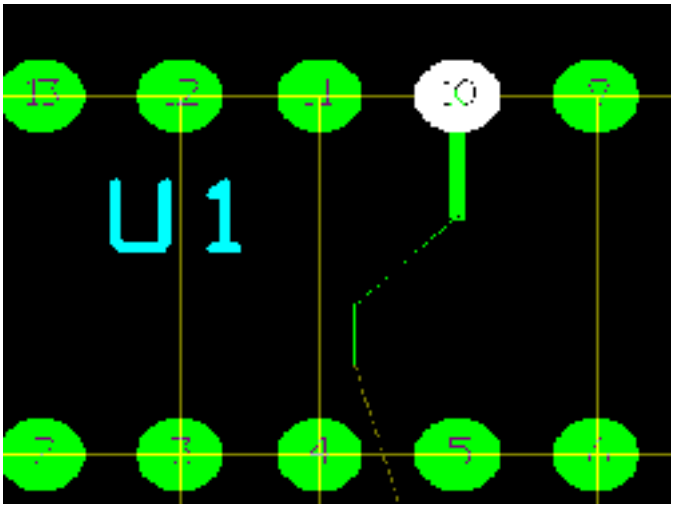
In the Route Settings dialog box you can choose to:

- never use hints, in which case the router uses its default algorithm
- use hints only when exiting pads and vias
- always use hints, which means that you can give Layout directional hints when exiting pads, vias, and also when continuing from the last routed segment endpoint.

The following illustration shows the temporary path of a connection after the router receives a directional hint that suggests continuing straight out from a routed segment endpoint.



The illustration below shows the temporary path of the same connection after a directional hint that suggests continuing from the endpoint diagonally.



Creating and modifying nets

In Layout, you can create nets manually using the connection tool.

Note: These modifications cannot be back annotated to the schematic design.

Creating nets

To create a net

1. Choose the connection toolbar button.
2. From the pop-up menu, choose Add.
3. Select a component pin.
4. Draw the new net and click the left mouse button on the end pad. The Modify Nets dialog box appears.
5. Enter the name of the new net, then choose the OK button.

6 From the pop-up menu, choose End Command.

Splitting nets

You can separate a net into two separate nets interactively.

To split a net

1 Choose the connection toolbar button.

2 From the pop-up menu, choose Delete.

3 On the board, select a net to split into two separate nets. (Do not select a pin at the end of a signal.) Layout asks you to confirm your decision to delete the connection.

4 Choose the Yes button. Layout asks if you're certain you want to split the net.

5 Choose the Yes button. The Modify Nets dialog box appears.

6 Enter the name of one of the new nets, then choose the OK button. The Modify Nets dialog box reappears.

7 Enter the name of the other new net, then choose the OK button.

Adding and deleting pins connected to nets

You can add and delete pins from nets on the board, or in the Nets spreadsheet.

To add or delete pins from a net

1 Choose the pin toolbar button.

2 Select a pin.

3 From the pop-up menu, choose Properties. The Modify Connections dialog box appears.

4 Select a new net name from the drop-down list, then choose the OK button.

Or

1. Open the Nets spreadsheet.
2. Select a net in the spreadsheet.
3. From the pop-up menu, choose Connection Edit. The Modify Connections dialog box appears.
4. Enter the names of the pins in the Pin list text box.
5. Select the Add option to add pins.
   or
   Select the Delete option to delete pins.
6. Choose the OK button.

Related topics

Nets spreadsheet

Disconnecting pins from nets

You can disconnect a pin from a net without splitting the net.

To remove a pin from a net

1. Choose the connection toolbar button.
2. From the pop-up menu, choose Disconnect Pin.
3. Select the pin. Layout asks you to confirm that you want to disconnect the pin.
4. Choose the Yes button. The pin is disconnected.

Generating test points

Layout provides great flexibility with test points. Because you can define one or more vias for use as test points, you can assign a distinctive shape or other characteristic to your test point vias.

You can generate test points automatically (Layout and Layout Plus) using the Autorouting tools, or you can place them interactively (all Layout editions) while you do Routing.

If you want a report of the test points you've created, choose Reports from the File menu, then select Test Points List and choose the OK button.

Note: In some cases, it is possible to create test points using routing vias, but this is not recommended. If you use routing vias as test points, then try to remove all test points, Layout will try to remove the routing vias that are doing double duty. For test points, you should always choose a via padstack that is not currently used for routing.

To generate test points automatically

1. In the Padstacks spreadsheet, select the first via that shows all layers undefined.
2. From the pop-up menu, choose the Properties command. The Edit Padstack dialog box displays.
3. Enable the Use for Test Point option and choose the OK button.
4. Define the shape and size of the test point via.
5. If you need additional vias available as test points, repeat the steps above.
6. In the Nets spreadsheet, select the nets that need test points.
7. From the pop-up menu, choose the Properties command. The Edit Net dialog box displays.
8. In the Net Attributes group box, select Test Point, then choose the OK Button.
9. Verify that the board is entirely routed.
10. From the Auto menu, point to Place and choose the Test Point command. The Test Point Settings dialog box displays. Make your choices and choose the OK button. Layout places test points on each of the specified nets.

To generate test points interactively

1. In the Padstacks spreadsheet, select the first via that shows all layers undefined.
2. From the pop-up menu, choose the Properties command. The Edit Padstack dialog box displays.
3. Enable the Use for Test Point option and choose the OK button.
4. Define the shape and size of the test point via.
5. If you need additional vias available as test points, repeat the steps above.
6. Select the manual routing tool.
7. Select the net that needs a test point, route it, then chose the Add Test Point command from the pop-up menu. Layout places the via and flags it as a test point.

Related topics

Add Test Point command

Test Point Settings dialog box

Checking routing

You should check the routing of a board using Route Spacing Violations, the density graph, and the routing information in the Statistics spreadsheet.

Using Route Spacing Violations

After you route the board, you should run Route Spacing Violations, which verifies adherence to spacing criteria as listed in the Route Spacing spreadsheet (choose the spreadsheet toolbar button, choose Strategy, then choose Route Spacing). Layout does not allow a spacing error to be created by the autorouter.

Any problem found by Route Spacing Violations is marked with a circle. You can find out the nature of the problem by choosing the query toolbar button, which brings up the query window. Then, when you choose the error toolbar button and select the error, the information about the error appears in the query window.

Note: For information on how to use the error tool to get more information about reported errors, see *Ensuring manufacturability*.

To use Route Spacing Violations

1. From the Auto menu, choose Design Rule Check. The Check Design Rules dialog box appears.
2. Choose the Clear All button.
3. Select the Route Spacing Violations option, then choose the OK button. Layout checks the board for route spacing violations and marks any errors with circles.

Viewing routing statistics

When you have finished routing the board, you can view the routing statistics in the Statistics spreadsheet. The spreadsheet gives the percentage and number of connections completed, via data, and more.

To view the routing statistics

1. Choose the spreadsheet toolbar button, then choose Statistics. The Statistics spreadsheet appears.
2. Scroll until you find the Routed row, which is the beginning of the routing data.
3. Close the spreadsheet when you are finished viewing the statistics.

Resistor packages (pin swapping)

The following methods can be used to optimize connectivity to resistor packages.

Option 1 optimizes each individual package for routing, while option 2 optimizes which package each connection will go to.

Resistor package pin swap option #1 (Preferred)---Automatic pin swapping within the same component using Swap Pins capabilities. This option is for optimization before routing and after placement.

Resistor package pin swap option #2 (Recommended only if necessary)---Automatic pin swapping between different components using Gate Swap capabilities. For placement optimization only (not for routing optimization).

Resistor package pin swap option #1

Once Swap Gates has taken place, and the connections are essentially attached to the correct components, a pin swap should take place. Pin Swaps are specifically designed to "untie" the connections among neighboring components, in order to solve in more detail the routing problems caused by convoluted connections.

To facilitate proper pin swapping, your packages (for components such as connectors and resistor packs) should be set up as follows:

1. Assign the same Gate Name for each of the pins that can be swapped (for example, "A").
2. All of the Pin Names should be different (for example, "LOAD1," "LOAD2," "LOAD3").
3. The Pin Group must be the same non-zero number for all pins (for example "Pin Group #1").
4. Enable a Pin Swap pass after the final Swap Component pass using Window/Place Pass. Note: This pass is disabled in the default strategies.

The sample of "RES_10SIP" shown below has 9 pins that can be swapped. The connections to each component using this Package description will be optimized for routing purposes (such things as connection "bowties" will be "untied"), but will not be swapped between components for placement purposes:

```
Pin 1,  Gate name: "VCC" Pin name: "VCC"   Gate group: 0 Pin Group: 0
Pin 2,  Gate name: "A"   Pin name: "LOAD1" Gate group: 1 Pin Group: 1
Pin 3,  Gate name: "A"   Pin name: "LOAD2" Gate group: 1 Pin Group: 1
Pin 4,  Gate name: "A"   Pin name: "LOAD3" Gate group: 1 Pin Group: 1
Pin 5,  Gate name: "A"   Pin name: "LOAD4" Gate group: 1 Pin Group: 1
Pin 6,  Gate name: "A"   Pin name: "LOAD5" Gate group: 1 Pin Group: 1
Pin 7,  Gate name: "A"   Pin name: "LOAD6" Gate group: 1 Pin Group: 1
Pin 8,  Gate name: "A"   Pin name: "LOAD7" Gate group: 1 Pin Group: 1
Pin 9,  Gate name: "A"   Pin name: "LOAD8" Gate group: 1 Pin Group: 1
Pin 10, Gate name: "A"   Pin name: "LOAD9" Gate group: 1 Pin Group: 1
```

Resistor package pin swap option #2

Layout does not swap pins between two different components (or gates) without a manual override. If you are working with components like resistor packages, and you can see that some pin swap between two or more components of the same Package type will help the design, you may want to use this technique.

1. Assign a different Gate Name for each of the pins that can be swapped (for example, "A" through "Z," or "AA" through "ZZ").
2. Be sure that the Pin Names for all of the pins are the same (for example, "LOAD").
3. Be sure that all of the gates are in the same Gate Group (for example, "Gate Group #1").
4. Enable Swap Gates during the normal Proximity Place pass in the default placement strategies, using Window/Place Pass. Note: This option is disabled in the default strategies.

The sample of "RES_10SIP" shown below has 9 pins that can be swapped. The connections to each component using this Package description will be swapped for placement

optimization, but convoluted connections will not necessarily be optimized for routing purposes:

```
Pin 1,  Gate name: "VCC" Pin name: "VCC"  Gate group: 0 Pin Group: 0
Pin 2,  Gate name: "A"   Pin name: "LOAD" Gate group: 1 Pin Group: 0
Pin 3,  Gate name: "B"   Pin name: "LOAD" Gate group: 1 Pin Group: 0
Pin 4,  Gate name: "C"   Pin name: "LOAD" Gate group: 1 Pin Group: 0
Pin 5,  Gate name: "D"   Pin name: "LOAD" Gate group: 1 Pin Group: 0
Pin 6,  Gate name: "E"   Pin name: "LOAD" Gate group: 1 Pin Group: 0
Pin 7,  Gate name: "F"   Pin name: "LOAD" Gate group: 1 Pin Group: 0
Pin 8,  Gate name: "G"   Pin name: "LOAD" Gate group: 1 Pin Group: 0
Pin 9,  Gate name: "H"   Pin name: "LOAD" Gate group: 1 Pin Group: 0
Pin 10, Gate name: "I"   Pin name: "LOAD" Gate group: 1 Pin Group: 0
```

Autorouting

Layout features two autorouting options: a gridded autorouter, and a gridless, shape-based autorouter.

Gridded autorouter

Layout's gridded autorouter has two key features: *sweep* technology, which allows you to specify the directional emphasis for routing different boards, and *shove* technology, which minimizes vias and allows extremely dense autorouting. In addition, you can use interactive routing tools (see *Interactive routing tools* below) with the gridded autorouter to refine the process of autorouting.

Note: For more information on Layout's gridded autorouter, see the *Layout Autorouter User's Guide.*

Sweep technology

The autorouter routes the board using *sweeps*, which are successive routing passes. Beginning at a specified point, Layout routes the board continually according to the sweep direction you specify. For example, if you want the sweeps to progress up first and then to the left, the autorouter routes vertically and then horizontally, working through the entire board.

Shove technology

The autorouter finds the optimal space for a given track and then moves or “shoves” other tracks out of the way before routing in that area. If a pad or via is blocking the routing path, then the autorouter attempts to go around the blockage by routing to a different layer using a via. The autorouter also checks to see if there are obstructing tracks that can be rerouted or moved to an entirely different location on the board.

Interactive routing tools

Though not part of the autorouter itself, interactive routing tools complement the gridded autorouter by allowing you to refine an autorouted board. Using auto path route mode and shove track mode, you can route critical nets and dense boards with minimal effort.

Gridless shape-based autorouter

Layout Plus provides a gridless shape-based autorouter called SmartRoute. Exclusive to Layout Plus, SmartRoute has fast routing speeds, high completion rates, and high router quality.

Note: For information on SmartRoute, see the *OrCAD Layout SmartRoute User’s Guide.*

Automatic routing using SPECCTRA

11

Overview

Layout allows you to interface directly with many of SPECCTRA's capabilities. You can run SPECCTRA from Layout to fanout selected components, route selected nets, miter wire corners in the completed board, or autoroute the entire design.

You can also launch SPECCTRA from Layout if you want to interactively route your board in SPECCTRA. After you complete routing the board in SPECCTRA, and exit SPECCTRA, the routing changes you made in SPECCTRA are displayed in Layout.

By default, SPECCTRA uses the .DO files that are generated using the settings you specify in the SPECCTRA Automatic Router Parameters dialog box to perform route actions. You can also use customized .DO files to autoroute your design using SPECCTRA.

Layout also allows you to generate placement and routing reports using SPECCTRA. These reports give you detailed information about the placement or routing characteristics of your PCB design, including conflicts or rule violations.

For more information on autorouting your board and generating placement and routing reports using SPECCTRA, see the following topics:

- Launching SPECCTRA from Layout on page 444
- Performing fanout using SPECCTRA on page 447
- Routing specific nets using SPECCTRA on page 448
- Mitering wire corners using SPECCTRA on page 449
- Autorouting the board using SPECCTRA on page 450
- Running SPECCTRA using a customized .DO file on page 451
- Generating SPECCTRA Reports on page 455

For more information on using SPECCTRA, see the following documentation:

- *Allegro PCB Router User Guide*
- *Allegro PCB Router Command Reference*

Prerequisites for automatic routing using SPECCTRA

This section describes the points you must note before routing your design using SPECCTRA.

- Ensure that you have specified all the options for routing your design in the SPECCTRA Automatic Router Parameters dialog box.

  To access the SPECCTRA Automatic Router Parameters dialog box, choose Autoroute SPECCTRA from the Auto menu, then choose Setup.

  Note: The settings in the SPECCTRA Automatic Router Parameters dialog box are saved in the LAYOUT.INI file and are available to all the projects on your system.

- Ensure that the route grid and via grid setting you specify in the Router Setup tab of the SPECCTRA Automatic Router Parameters dialog box is the same or a multiple of the route grid and via grid setting you specify in the System Settings dialog box.

- Ensure that you have enabled routing for all the nets you want to be routed in SPECCTRA.

To enable routing for nets, do the following:

a. Select the nets for which you want to enable routing in the Nets spreadsheet.

b. From the Edit menu, choose Properties.

The Edit Net dialog box appears.

c. Select the Routing Enabled check box and click OK.

The text Yes in the Routing Enabled column in the Nets spreadsheet indicates that routing is enabled for a net.

- Before you autoroute your board using SPECCTRA, you must remove the unused footprints, padstacks, and nets on the board by doing the following:

a. From the Auto menu in Layout, choose Cleanup Design.

The Cleanup Design dialog box appears.

b. Deselect all the check boxes in the Cleanup Routing group box.

c. Select the following check boxes:

- Remove Unused Padstacks
- Remove Unused Footprints
- Remove Unused Nets

d. Click OK.

- Place route keepouts around copper pours before autorouting the board using SPECCTRA. If you do not do this, SPECCTRA will route through the copper pours.
- When you autoroute your design using SPECCTRA, a keepout region is created around any component that has the Not in Netlist attribute. The Not in Netlist attribute is typically set for non-electrical parts, such as mounting holes. Check component properties to ensure that the Not in Netlist attribute has not been unintentionally set for components.

To check component properties, do the following:

a. Choose the Spreadsheets toolbar button.

b. Select Components from the drop-down list.

c. Double-click on a component.

The Edit Component dialog box appears displaying the properties for the component.

- All padstacks must have a size and shape definition on at least one layer. If necessary, you can define a very small pad size, but you cannot leave it completely undefined. Use the Padstack spreadsheet in Layout to identify undefined pads, if any.
- Do not use SPECCTRA to fanout a component on a board having split planes if one of the nets connected to the component is connected to a split plane. Use Layout's fanout command to fanout components on boards having split panes.
- SPECCTRA does not support positive shapes (copper pours on routing layers). As a result, the copper pours in your board will be lost if you route your board using SPECCTRA.
- SPECCTRA does not support spaces in file names and in file paths. Do not use spaces in the board file name, in the .DO file name, and in the path to the board file or the .DO file.

Launching SPECCTRA from Layout

You can open a board design in Layout and launch SPECCTRA from Layout if you want to interactively route the board in SPECCTRA.

When you launch SPECCTRA from Layout, the design is read into SPECCTRA. You can then interactively route the design using SPECCTRA. After you complete routing the design in SPECCTRA, and exit SPECCTRA, the routing changes you made in SPECCTRA are displayed in Layout.

Before you launch SPECCTRA from Layout, you must specify the location of the SPECCTRA installation you want to use for routing your board design.

Note: If you have previously run SPECCTRA from Layout, Layout remembers the most recently used version of SPECCTRA.

To specify the location of the SPECCTRA installation

1. From the Auto menu, choose Autoroute SPECCTRA, then choose Setup.

   The SPECCTRA Automatic Router Parameters dialog box appears.

2. Click the General tab.

3. Enter the path to the SPECCTRA.EXE file in the SPECCTRA installation that you want to use for routing your board, or click Browse to select the SPECCTRA.EXE file.

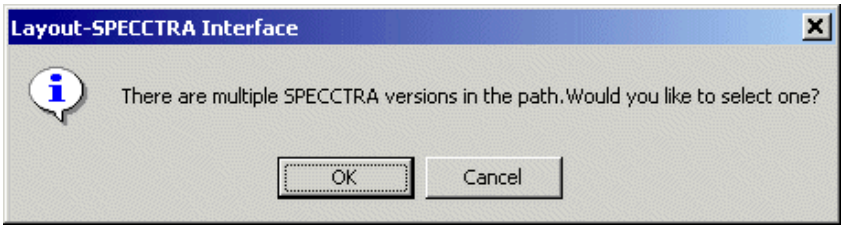
   If you click the Browse button, the Find SPECCTRA for you? message box appears.

Click	To
Yes	Display the Find SPECCTRA dialog box. The Find SPECCTRA dialog box displays the versions of SPECCTRA.EXE found in the directories listed in the PATH environment variable. Do one of the following: ■ Select the SPECCTRA version you want to use and click Select. ■ Click Cancel to manually browse for the SPECCTRA.EXE file.
No	Manually browse for the SPECCTRA.EXE file.

To launch SPECCTRA from Layout

1 From the Auto menu, choose Autoroute SPECCTRA, then choose Launch SPECCTRA.

If you did not specify the location of the SPECCTRA installation you want to use to route your board, the following message box appears:



Click OK to display the Find SPECCTRA dialog box.

The Find SPECCTRA dialog box displays the versions of SPECCTRA.EXE found in the directories listed in the PATH environment variable.

Do one of the following:

- Select the SPECCTRA version you want to use and click Select.
- Click Cancel to manually browse for the SPECCTRA.EXE file.

2 The board design you opened in Layout is read into SPECCTRA. You can now interactively route the design using SPECCTRA.

Note: While your design is opened in SPECCTRA, you cannot make any changes in Layout.

3 After you complete routing the design in SPECCTRA, choose Quit from the File menu.

The Save and Quit dialog box appears.

4 Do one of the following:

- Click Save and Quit if you want to save the changes you made in SPECCTRA. The board is automatically imported into Layout. The changes you made in SPECCTRA are displayed in Layout.

- Click Quit if you do not want to save the changes you made in SPECCTRA. The board is automatically opened in Layout without the changes you made in SPECCTRA.

Performing fanout using SPECCTRA

Layout allows you to fanout specific components in your design using SPECCTRA.

When you fanout a component, SPECCTRA routes short escape wires from pads to vias. Routing short escape wires from pads to vias allows subsequent routing of these connections on additional layers.

SPECCTRA chooses the escape vias from the available via set and places them on the current via grid.

Before you perform fanout using SPECCTRA, you must specify the options for performing fanout.

To specify fanout options

1. From the Auto menu, choose Autoroute SPECCTRA, then Setup.

   The SPECCTRA Automatic Router Parameters dialog box appears.

2. Click the Fanout tab and specify the fanout options.

Tip

You can specify fanout options using a custom .DO file. For more information, see Running SPECCTRA using a customized .DO file on page 451.

To fanout components

1. From the Auto menu, choose Autoroute SPECCTRA, then Fanout.

The Components spreadsheet appears.

2 Select the components you want to fanout. To select multiple components, press the CTRL or Shift key and click on the components you want to fanout.

You can also select the components you want to fanout from the Layout design window. To do this, minimize the Components spreadsheet and select components from the Layout design window.

3 Click the right mouse button and choose Fanout Using SPECCTRA from the shortcut menu.

The Layout-SPECCTRA Interface Status dialog box appears displaying the progress of the fanout process. Click Cancel if you want to cancel the fanout process.

Note: When you perform fanout using SPECCTRA, SPECCTRA is run in the background to fanout the selected components.

Routing specific nets using SPECCTRA

You can route specific nets in your design using SPECCTRA and do the rest of the routing in Layout.

After routing the nets in SPECCTRA, you can lock the routed nets so that they are not rerouted by Layout.

Before you route nets using SPECCTRA, you must specify the routing options.

To specify options for routing nets

1 From the Auto menu, choose Autoroute SPECCTRA, then Setup.

The SPECCTRA Automatic Router Parameters dialog box appears.

2 Specify the options for routing nets in the Router Setup and Route Strategy tabs.

Tip

You can specify the options for routing nets using a custom .DO file. For more information, see Running SPECCTRA using a customized .DO file on page 451.

To route specific nets using SPECCTRA

1. From the Auto menu, choose Autoroute SPECCTRA, then Route Net/Nets.

   The Nets spreadsheet appears.

2. Select the nets you want to route. To select multiple nets, press the CTRL or Shift key and click on the nets you want to route.

3. Click the right mouse button and choose Route Using SPECCTRA from the shortcut menu.

   The Layout-SPECCTRA Interface Status dialog box appears displaying the progress of the routing process. Click Cancel if you want to cancel the routing process.

Note: When you route nets using SPECCTRA, SPECCTRA is run in the background to route the selected nets.

Tip

You can lock the routed nets so that they are not rerouted by Layout. For information on locking routed nets, see Locking routed tracks on page 421.

Mitering wire corners using SPECCTRA

Layout allows you to miter wire corners in the completed board using SPECCTRA.

SPECCTRA changes all the 90 degree wire corners in the design to 45 degrees.

To miter wire corners using SPECCTRA

1 From the Auto menu, choose Autoroute SPECCTRA, then Miter Corner.

The Layout-SPECCTRA Interface Status dialog box appears displaying the progress of the miter corner process. Click Cancel if you want to cancel the miter corner process.

All the 90 degree wire corners in the design are changed to 45 degrees.

Note: When you run miter corner using SPECCTRA, SPECCTRA is run in the background to miter the wire corners in the design.

Autorouting the board using SPECCTRA

Layout allows you autoroute the board you have opened in Layout using SPECCTRA.

Before you autoroute the board **To specify options for autorouting the board**

1 From the Auto menu, choose Autoroute SPECCTRA, then Setup.

The SPECCTRA Automatic Router Parameters dialog box appears.

2 Specify the options for autorouting the board in the Fanout, Router Setup and Route Strategy tabs.

Tip

You can specify autorouting options using a custom .DO file. For more information, see Running SPECCTRA using a customized .DO file on page 451.

To autoroute the board using SPECCTRA

1 From the Auto menu, choose Autoroute SPECCTRA, then Route Board.

The board design you opened in Layout is read into SPECCTRA, and SPECCTRA starts autorouting the design. When SPECCTRA completes autorouting the design the Save and Quit dialog box appears.

2 Do one of the following:

- Click Save and Quit, if you want to save the results of the autorouting process. The autorouted board is automatically imported into Layout.
- Click Quit, if you do not want to save the results of the autorouting process. The board is automatically opened in Layout without the changes.

Running SPECCTRA using a customized .DO file

You can perform fanout, route net and autorouting operations by running SPECCTRA from Layout using a customized .DO file.

To run SPECCTRA using a customized .DO file

1 From the Auto menu, choose Autoroute SPECCTRA, then Setup.

The SPECCTRA Automatic Router Parameters dialog box appears.

2 Click the General tab.

3 Select the Use a customized DO file check box.

4 Enter the path and name of the customized .DO file you want to use, or click the browse button to select the .DO file.

For information on using your own .DO file or creating a default .DO file for running SPECCTRA from Layout, see the following topics:

- To use your own .DO file for running SPECCTRA from Layout on page 452
- To create a default .DO file for running SPECCTRA from Layout on page 452
- To customize the .DO files generated by SPECCTRA on page 453

Important

When you run SPECCTRA using a customized .DO file, the settings in the SPECCTRA Automatic Router Parameters dialog box will be ignored.

To use your own .DO file for running SPECCTRA from Layout

1. Open your .DO file in a text editor such as Notepad.
2. Add one of following commands to the end of the file:

   ```
   write routes $/SpecctraWithinLayout.rte
   ```

   OR

   ```
   write routes $/SpecctraWithinLayout.ses
   ```

   This command tells SPECCTRA to write the route (RTE) or session (SES) file in the same directory as your board. The file must be named SpecctraWithinLayout.RTE or SpecctraWithinLayout.SES so that Layout can automatically reload your design after you close SPECCTRA.

To create a default .DO file for running SPECCTRA from Layout

1. Click the Create button in the General tab of the SPECCTRA Automatic Router Parameters dialog box.

   The SPECCTRA template DO file dialog box appears.
2. Enter the name of the .DO file in the File name field and click Save.

A template .DO file with the default options that you can use to run SPECCTRA from Layout is created.

3 In the Would you like to edit the new DO file dialog box, click Yes to edit the new file with Notepad.

There are many commands that are included, but not enabled in the default .DO file. To enable these commands, remove the comment (#) from the respective command line. Do not remove the following command from the .DO file.

```
write routes $/SpecctraWithinLayout.rte
```

This command tells SPECCTRA to write the route (RTE) file in the same directory as your board. The file must be named SpecctraWithinLayout.RTE so that Layout can automatically reload your design after you close SPECCTRA.

4 Save the changes you make and exit Notepad.

Note: You can use the command:

```
write routes $/SpecctraWithinLayout.ses
```

instead of the command:

```
write routes $/SpecctraWithinLayout.rte
```

in the .DO file, if you want SPECCTRA to write the session (SES) file in the same directory as your board. Layout reads the .RTE or .SES file having the latest time stamp.

To customize the .DO files generated by SPECCTRA

When you run SPECCTRA from Layout without using a customized .DO file, the following .DO files are generated in the directory where the .MAX file is located. These .DO files are generated using the settings in the SPECCTRA Automatic Router Parameters dialog box.

FANOUTCOMP.DO	This file is created when you perform fanout using the procedure described in Performing fanout using SPECCTRA on page 447.

MITERCORNER.DO	This file is created when you run miter corner using the procedure described in Mitering wire corners using SPECCTRA on page 449.
ROUTEBOARD.DO	This file is created when you autoroute the board using the procedure described in Autorouting the board using SPECCTRA on page 450.
ROUTENET.DO	This file is created when you route nets using the procedure described in Routing specific nets using SPECCTRA on page 448.

You can customize the generated .DO files, by doing the following:

1. Copy the generated .DO file to a different file name.

   This is required because the generated .DO files are overwritten every time you run SPECCTRA from Layout.

2. Edit the renamed .DO file.

   Note: Do not remove the following command from the .DO file.

   ```
   write routes $/SpecctraWithinLayout.rte
   ```

   This command tells SPECCTRA to write the route (RTE) file in the same directory as your board. The file must be named SpecctraWithinLayout.RTE so that Layout can automatically reload your design after you close SPECCTRA.

Note: You can use the command:

```
write routes $/SpecctraWithinLayout.ses
```

instead of the command:

```
write routes $/SpecctraWithinLayout.rte
```

in the .DO file, if you want SPECCTRA to write the session (SES) file in the same directory as your board. Layout reads the .RTE or .SES file having the latest time stamp.

Generating SPECCTRA Reports

You can generate placement and routing reports using SPECCTRA from Layout. These reports give you detailed information about the placement or routing characteristics of your PCB design, including conflicts or rule violations.

The reports generated using SPECCTRA are created in the project directory.

To generate a report using SPECCTRA

1. From the Auto menu, choose Autoroute SPECCTRA, then View Report.

   The SPECCTRA Reports dialog box appears. The Reports Type list displays the reports you can generate using SPECCTRA.

2. Select the report you want to generate from the Reports Type list.

3. Click the Add button to add the selected report to the list on the right hand side. You can select multiple reports and add them at the same time. The list on the right hand side displays the list of reports to be generated.

   If you do not want to generate a report, select it in the list on the right hand side, and click the Remove button to remove the report from the list of reports to be generated. You can select multiple reports and remove them at the same time.

4. Select the Status of last SPECCTRA operation check box if you want to view a report displaying the status of the last operation SPECCTRA performed on your board.

   For example, after running a fanout operation using SPECCTRA, if you select this check box and click OK, SPECCTRA displays a report named MONITOR.STS displaying the status of the fanout operation.

5. Click OK to generate the selected reports.

   The reports are created in the project directory and displayed in a text editor.

To generate a report using a customized .DO file

1. From the Auto menu, choose Autoroute SPECCTRA, then View Report.

   The SPECCTRA Reports dialog box appears.

2. Select the Use a Customized DO File check box.

3. Enter the name and path to the .DO file or click the Browse button to select the file.

   Note: Ensure that the last line in the .DO file has the following command:

   ```
   quit -c
   ```

   To edit the .DO file, click the Edit button.

4. Click OK to generate reports using the .DO file.

   The reports are created in the project directory and displayed in a text editor.

Using thermal reliefs and copper pour zones

12

This chapter explains how to use thermal reliefs and copper pour zones.

Thermal reliefs

Thermal relief pads are used as contacts to plane layers or copper zones, for applications such as the connection to power and ground on a multilayer board.

There are two things you must do before defining thermal reliefs:

- designate the target layer for the thermal reliefs as a plane layer in the Layers spreadsheet
- assign a net to the layer.

Note: For information on designating plane layers, defining vias, and setting net properties, see *Chapter 5, "Setting up the board."*

Note: When viewing a plane layer, the background represents copper, and the foreground represents cleared areas.

Defining thermal reliefs

You can specify *relative* dimensions for small and large thermal reliefs by editing the default values in the Thermal Relief Settings dialog box. The dimension options include the sizes for annular over drill, isolation width, and spoke width.

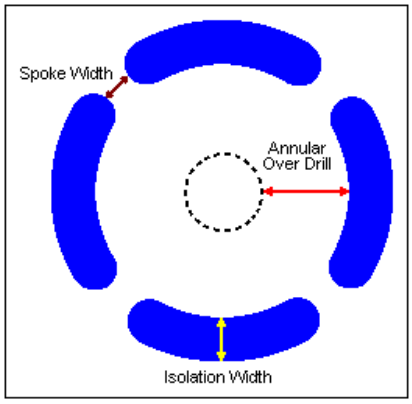


Figure 12-1 A thermal relief pad.

Small thermal reliefs are used throughout the board by default. You can assign large thermal reliefs to a particular padstack using the Edit Padstack dialog box (accessed by double-clicking in a cell in the Padstacks spreadsheet).

To specify dimensions for the thermal reliefs

1. From the Options menu, choose Thermal Relief Settings. The Thermal Relief Settings dialog box appears.
2. Edit the settings for the following options in both the Small Thermal Relief group box and the Large Thermal Relief group box, then choose the OK button.

   Annular over drill After drilling, the width remaining between the drilled hole and the inside of the isolation ring.

   Isolation Width The width of the isolation ring that surrounds the pad.

   Spoke Width The width of the copper tie that connects the pad to the plane.

Note: The spoke width value specified in the Thermal Relief Settings dialog box is used for copper pour, as well as for plane layers.

To assign large thermal reliefs

1. Choose the spreadsheet toolbar button, then choose Padstacks. The Padstacks spreadsheet appears.
2. Double-click on the name of the padstack to which you want to assign a large thermal relief. The Edit Padstack dialog box appears.
3. Select the Large Thermal Relief option, then choose the OK button. Layout assigns a large thermal relief to the padstack. It will have the relative dimensions that you specified in the Thermal Relief Settings dialog box.

Previewing thermal reliefs

You can preview thermal reliefs to check their connections to the board.

To preview thermal reliefs

1. In the design window, press the BACKSPACE key. Layout displays a blank screen.
2. Type the number that corresponds to the layer that you want to view (for example, 3 for the ground layer). Layout draws just that layer.
3. View the thermal connections.

4 To return to the previous design view, press F5.

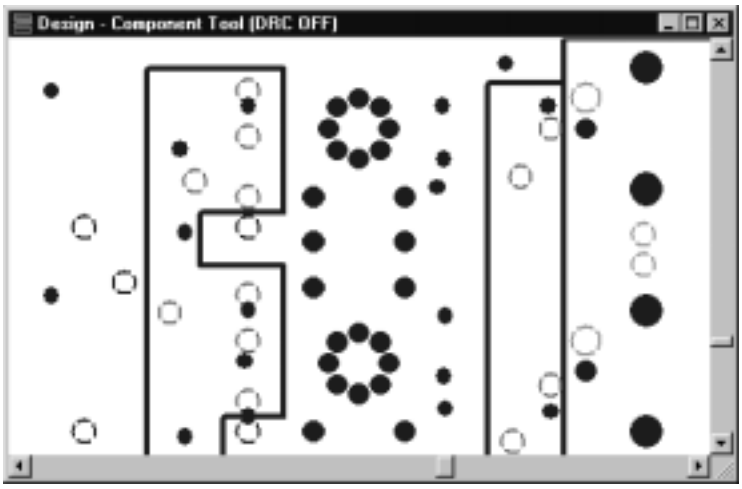


Figure 12-2 Previewing thermal reliefs.

Rules that apply to creating thermal reliefs

Layout follows the rules below to determine which pads are assigned thermal reliefs on the plane layers and in what order.

- If the entire net is unrouted, all through-hole pads attached to nets are assigned a thermal relief.
- Routed sections of nets are considered *subnets*. Each subnet must have at least one thermal relief. Subnets employ the following search order for assigning a thermal relief.

1 Vias are always assigned thermal reliefs. For example, if you route between a capacitor on the bottom of the board and an IC on the top of the board, the via will have a thermal relief.

2 If the subnet does not find a via, any pad marked as a forced thermal relief becomes the thermal relief for that subnet.

3 If the subnet does not find a via or a pad marked as a forced thermal relief, the first pad marked as a preferred thermal relief becomes the thermal relief for that subnet.

4 If the subnet does not find a via or a pad marked as a forced or preferred thermal relief, global or standard pads receive thermal reliefs.

5 If the subnet does not find a via, a forced or preferred thermal relief, or a global or standard pad, the pad for the thermal relief is picked at random.

6 If no pad fits the correct criteria, a design rule check for dispersion creates an error at each pad that fails to connect to the plane.

Note: SMD pads cannot connect to a plane using thermal reliefs. If you are using Layout or Layout Plus, see Fanout on boards with surface mount devices in the *OrCAD Layout Autorouter User's Guide*. If you are using Layout Engineer's Edition, see Routing power and ground in Chapter 8, Routing the board. Forced thermal reliefs and preferred thermal reliefs

Forced thermal reliefs and preferred thermal reliefs

If you designate a footprint pad as a forced thermal relief, then as long as the pad is attached to the appropriate net, the pad is assigned a thermal relief on the plane layers that are attached to that net.

If you designate a footprint pad as a preferred thermal relief, then as long as the pad is attached to the appropriate net, the pad will be the first in each subnet (routed portion of the net) to be assigned a thermal relief on the plane layers that are attached to that net. If there is already a via on the subnet, the via will receive a thermal relief, because vias are always assigned thermal reliefs.

To designate a pad as a forced or preferred thermal relief

1 Choose the spreadsheet toolbar button, then choose Footprints. The Footprints spreadsheet appears.

2 Select the footprint pad that you want to designate as a forced or preferred thermal relief, then choose Properties from the pop-up menu. The Edit Pad dialog box appears.

3 Select the Forced Thermal Relief option.
or
Select the Preferred Thermal Relief option.

4 Choose the OK button. Layout designates the pad as either a forced thermal relief or as a preferred thermal relief.

Using padstacks to create thermal reliefs

You can also assign thermal reliefs using the Edit Padstack dialog box. In the dialog box, you can assign a thermal relief to any pin independent of its net designation. The thermal reliefs assigned in this dialog box are forced thermal reliefs, and override preferred thermal reliefs as specified in the Edit Footprint dialog box.

Note: By default, Layout assigns thermal reliefs to nets connected to plane layers. You can use the command described here to connect a pin to the plane layer regardless of its net assignment.

To create thermal reliefs using padstacks

1 Choose the spreadsheet toolbar button, then choose Padstacks. The Padstacks spreadsheet appears.

2 Double-click on the layer you want to edit. The Edit Padstack Layer dialog box appears.

3 In the Pad Shape group box, select the Thermal Relief option, then choose the OK button. Layout assigns a thermal relief to the padstack. When the padstack is assigned to a pin, the thermal relief will be forced on that pin, regardless of net or thermal preference.

Copper pour zones

A copper pour zone is used to place copper in designated areas. It also places thermal reliefs on pads, while preventing copper islands. You create a copper pour zone by drawing and modifying an obstacle. A copper pour outline can be any

shape, using angles and arcs as needed. It can be attached to a component pin. Copper that is attached to a net assumes the properties of that net.

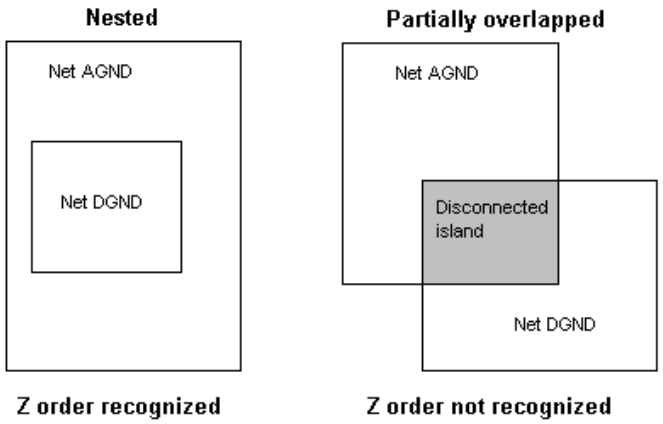
Note: In order to use copper pour, you must select the Enable Copper Pour option in the User Preferences dialog box. For more information, see Setting environment preferences.

Note: In the User Preferences dialog box, ensure that the Enable Copper Pour option is selected before you create a Gerber plot. Otherwise, your Gerber plots will have no copper pour in them.

A copper pour zone can be placed on any layer, can be solid or cross-hatched, and can be attached to any net. The hatch pattern is set in the Hatch Pattern dialog box (choose the Hatch Pattern button in the Edit Obstacle dialog box). The cross-hatch can be at any angle that is a multiple of 45˚.

You can also create nested copper pours on a plane and assign them a Z order. Layout nests a higher-numbered Z order pour within the next lower-numbered Z order pour, and applies the applicable clearance rules.

Note: Layout does not recognize Z order with copper pours that partially overlap. Incomplete overlapping causes a disconnect-island between the overlapping pours.



There are three types of obstacles in Layout that you need to be aware of when working with copper:

- *Anti-copper.* Use anti-copper to create non-copper areas within copper pour.
- *Copper area.* You can use copper areas to create custom pad shapes or other copper areas in which vias can't be placed or in which routing cannot occur. Isolation rules do not apply to copper areas.
- *Copper pour.* Copper pour obeys the isolation rules assigned to tracks with the same net as the copper pour. This includes any layer-specific clearances you have created for your nets. For instance, the copper pour clearance around a pad is the same as the track-to-pad clearance for that net.

Note: The spoke width value defined in the Thermal Relief Settings dialog box is used for copper pour, as well as for plane layers. For information on editing this value, see Defining thermal reliefs.

Note: After you've created copper pour, you can use the Use Fast Fill Mode option in the User Preferences dialog box (from the Options menu, choose User Preferences) to reduce screen redrawing time for copper pour fill. You must disable this option, however, before you create a Gerber plot.

Designating a seed point

If you want to use the Seed only from designated object option in the Edit Obstacle dialog box (see *Creating a copper pour zone* in this chapter), you have to designate a seed point. The seed point is the pad from which the copper pours.

To designate a seed point

1. Choose the pin toolbar button.
2. Select a pin that is attached to the net to which you want to attach the copper pour zone.
3. From the pop-up menu, choose Toggle Copper Pour Seed. Layout marks the pin with an "X," to indicate that

the pin is the copper pour seed point from which the copper will pour.

Creating a copper pour

This section explains how to create a typical copper pour, create a circular copper pour, specify a hatch pattern, and repour the copper after modifying the board.

To create a copper pour

1. Choose the obstacle toolbar button.
2. From the pop-up menu, choose New.
3. Click the left mouse button and drag to create the area that you want to designate as a copper pour.
4. Press the CTRL key and the left mouse button to select the obstacle. The cursor changes to a small cross.
5. From the pop-up menu, choose Properties. The Edit Obstacle dialog box appears.
6. From the Obstacle Type drop-down list, select Copper pour.
7. From the Obstacle Layer drop-down list, select an appropriate layer.
8. In the Copper Pour Rules group box, specify the following.

   Note: Here are some rules of thumb for setting copper pour rule options:

   - If you don’t select the Seed only from designated object option or the Isolate all tracks option, the copper pour seeds from all pads, vias, tracks, and net-attributed obstacles with the same net as the copper pour. Copper pour flows over tracks and vias belonging to the same net.
   - If you select the Seed only from designated object option, but not the Isolate all tracks option, the copper pour seeds only from pads marked as seeds.

Pour flows over tracks and vias belonging to the same net.

- If you select both the Seed only from designated object option and the Isolate all tracks option, the copper pour seeds only from pads marked as seeds. Copper pour is isolated from all tracks, even if they belong to the same net as the copper pour. Selecting both options is typically only done when you want to use the copper pour to create an EMI shield.

Clearance Designates the absolute clearance between this particular piece of copper pour and all other objects. A clearance of zero designates that the default clearances from each type of object will be used.

Z order Specifies the priority of the copper pour when it is nested with another copper pour. The higher the z-order value, the higher priority the copper pour has over other copper pours at the same location. For example, imagine you are looking down on the layer from above it. Copper pours with a higher z-order value sit above the lower ones and own the overlapping regions. The appropriate clearance between the copper pours is automatically maintained for you.

Isolate all tracks Normally, copper pour flows over tracks and vias belonging to the same net as the copper pour. By selecting this option, all tracks and vias are isolated from the copper pour, regardless of their net.

Seed only from designated object Normally, copper pour seeds from all tracks, vias, and pads belonging to the same net as the copper pour. By selecting this option, only pads marked as seed points will seed the copper pour. If you are creating an EMI shield, select both the Isolate all tracks option and the Seed only from designated object option, then designate a centrally located pad as your seed point.

Note: If you want to force the vias to be connected to the copper pour only through thermal spokes, edit the line in the [LAYOUT_GLOBALS] section of LAYOUT.INI to read:
THERMAL_COPPER_POUR_VIAS=YES
Without this modification, vias on the same net as the copper pour are flooded with copper.

9 If desired, select a net to attach to the copper pour from the Net Attachment drop-down list.

10 Choose the OK button. The copper pour is drawn on the screen.

Creating a circular copper pour

To create a circular copper pour

1 Designate a seed point. (See *To designate a seed point* earlier in this chapter.)

2 Choose the obstacle toolbar button.

3 From the pop-up menu, choose New.

4 From the pop-up menu, choose Properties. The Edit Obstacle dialog box appears.

5 From the Obstacle Type drop-down list, select Copper pour.

6 From the Obstacle Layer drop-down list, select an appropriate layer.

7 Specify other settings in the dialog box as necessary, then choose the OK button.

8 Click the left mouse button at the desired center for the circular copper pour.

9 From the pop-up menu, choose Arc.

10 Drag the cursor to create a circle of the desired size, then click the left mouse button to stop drawing. The copper pour forms on the screen.

Specifying a hatch pattern

To specify a hatch pattern for a copper pour

1 Double-click on an obstacle. The Edit Obstacle dialog box appears.

2 Choose the Hatch Pattern button. The Hatch Pattern dialog box appears.

3 Specify the settings as desired, then choose the OK button.

- **Line** Specifies the pattern as straight lines.
- **Cross Hatching** Specifies the pattern as crossed lines.
- **Solid** Specifies the pattern as solid pour. When you select Solid, the Hatch Grid setting is ignored and the grid is set to 90% of the Width value in the Edit Obstacle dialog box.
- **Hatch Grid** Specifies the spacing between the lines in the pattern.
- **Hatch Rotation** Specifies the angle of the lines in the pattern. Only angles in increments of 45˚ are supported.

4 Choose the OK button to close the Edit Obstacle dialog box. Layout draws the copper pour with the hatch pattern you specified.

Note: The more complex the hatch pattern, the slower the copper will pour. For example, hatch patterns that are not either horizontal or vertical pour quite slowly. For this reason, you should avoid small grid, cross hatching patterns at odd angles of rotation.

Refreshing a copper pour after editing the board

To refresh copper pour after editing the board

1 Edit the board as necessary.

2 From the Options menu, choose User Preferences. The User Preferences dialog box appears.

3 Ensure that the Enable Copper Pour option is selected, then choose the OK button.

Note: In the User Preferences dialog box, you can select

the Use Fast Fill Mode option to accelerate redrawing copper pour. For more information, see Setting environment preferences.

4 Choose the refresh all toolbar button. Layout repours the copper. The pour area adjusts automatically to accommodate your board edits.

Using copper pour as a shield

To create copper pour that serves as a shield

1 Choose the Obstacle Tool toolbar button.

2 From the pop-up menu, choose New.

3 From the pop-up menu, choose Properties. The Edit Obstacle dialog box displays.

4 Select Copper pour in the Obstacle Type drop-down list box.

5 Select the copper pour layer in the Obstacle layer drop-down list box.

6 Enter the copper pour netname in the Net Attachment text box. (The name should be entered in capital letters.)

7 Select the two options to Seed only from designated object and to Isolate all routes.

8 Choose the OK button to close the Edit Obstacle dialog box.

9 Press and hold the left mouse button to draw the polygon, then from the pop-up menu, choose the Finish command to accept the shape.

10 Choose the Pin Tool on the toolbar.

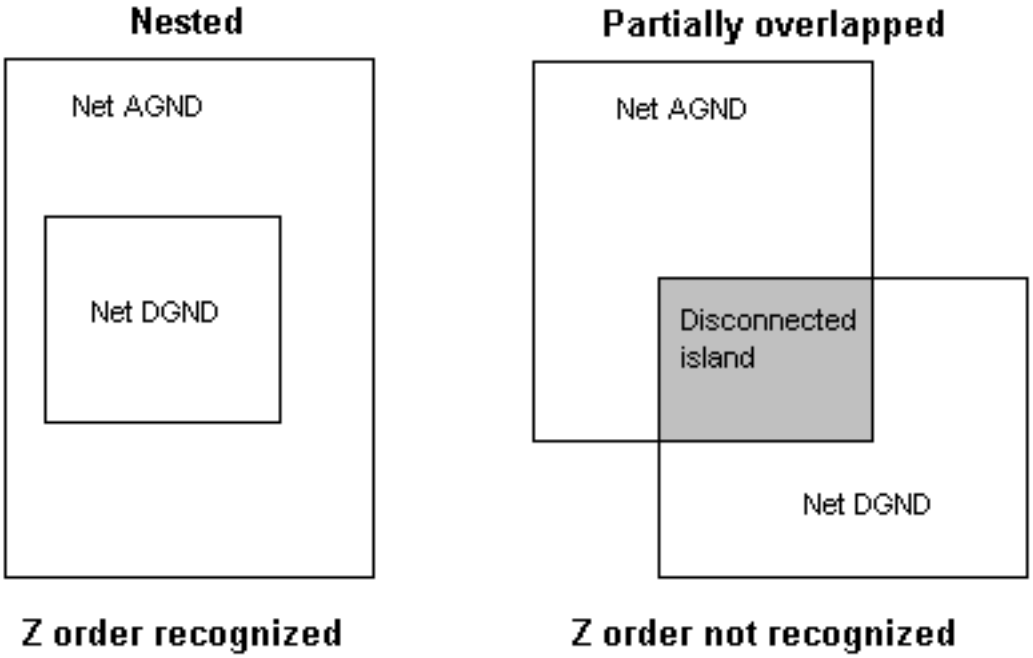
11 Select a pad that belongs to the same net as the copper pour.

12 From the pop-up menu, choose Toggle Copper Pour Seed. The pad becomes a seed point for the copper pour, designated by an "X" that displays on the pad.

13 Choose the Refresh Copper Pour toolbar button.

Note: In the User Preferences dialog box, you must keep the Enable Copper Pour option selected, but deselect the Use Fast Fill Mode option before you create a Gerber plot. Otherwise, your Gerber plots will have no copper pours.

Note: On a plane layer, Layout recognizes Z order with nested copper pours, but not with pours that partially overlap. Incomplete overlapping causes a disconnect-island between the overlapping pours.



Related topics

Copper pour zones

Refreshing a copper pour after editing the board

Hatch Pattern dialog box

Creating a copper pour

Ensuring manufacturability

13

This chapter explains the steps you should take to ensure that your board can be manufactured. They include:

- checking design rules
- investigating errors
- removing violations
- cleaning up your design

Checking design rules

Running the Design Rule Check command tests the integrity of your board by verifying the board's adherence to design rules.

To check design rules

1 From the Auto menu, choose Design Rule Check. The Check Design Rules dialog box appears.

Note: The DRC check toolbar button runs Design Rule Check with whichever options are selected in the Check Design Rules dialog box.

2 Select from the following options, then choose the OK button. Layout performs the specified checks and marks the errors with circles on the board.

Placement Spacing Violations Looks for component-to-component spacing violations and components that violate height restrictions, insertion outlines, or grid restrictions.

Route Spacing Violations Verifies adherence to spacing criteria listed in the Route Spacing spreadsheet.

Net Rule Violations Checks for any net parameters that are outside the rules listed in the Nets spreadsheet.

Copper Continuity Violations Checks for net-attached copper that is either attached to the wrong net, or not attached to its net.

Via Location Violations Checks for vias that violate any via location rules.

Pad Exit Violations Checks for routing that does not adhere to the pad exit criteria listed in the Footprints spreadsheet.

SMD Fanout Violations Checks for any enabled nets that come from SMD pads and do not terminate at either a through-hole or a via.

Test Point Violations Verifies that each net enabled for a test point actually has a test point.

Investigating errors

When you run Design Rule Check, the errors are marked on the board with circles. You can query an error to receive a full description of the problem.

Note: You can also view the errors in the Error Markers spreadsheet. To remove errors, select them all by clicking in the Location header cell in the Error Markers spreadsheet, then press the DELETE key.

To query errors

1. Choose the query toolbar button. The query window appears.
2. Choose the error toolbar button.
3. Select an error circle. A description of the error appears in the query window.
4. Take the necessary action to reconcile the error.

Note: When you move the pointer into the query window, its shape changes to a “Q,” to indicate that you can click on a keyword (any word enclosed in quotation marks) to get additional information.

Handling Known Errors

While designing a board, you may come across conditions that violate one or more design rules, but are acceptable and should not be considered as errors.

To handle such situations where the DRC errors are acceptable and should not appear as errors, you need to perform the steps listed below:

1. Run design rules check.
2. From the error report, select acceptable errors and right-click.
3. From the pop-up menu, choose Mark as Good DRC.

   For the errors marked as Good DRC, the marker color changes both on the Error Markers spreadsheet as well as on the layout board.

You can also mark a DRC error as known good error in the layout board itself. To do this:

1. Select the Error Tool.
2. Select the DRC error markers for acceptable errors.
3. Right-click and from the pop-up menu, choose Mark as Good DRC.

The error will be marked as Good DRC and the marker color will change. If required, you can change the color of the known good DRC marker. To know more about changing good DRC marker colors, see Modifying the color of known good DRC marker.

If you now check the Statistics spreadsheet, the errors marked as Good DRCs appear in the total error count but are not considered while calculating total enabled errors.

If you now run the design rule check again, the errors marked as good DRCs will also get deleted and appear as errors in the new error report.

You can prevent deleting of Good DRCs by following the steps listed below.

1. From the Auto drop-down menu, choose Design Rule Check.
2. In the Check Design Rules dialog box, select the Do not Delete Known Good DRC check box and click OK to save your changes.

If you now run the design rule check, the errors marked as Good DRCs will not be deleted unless you explicitly delete these.

Hiding known good DRC errors

If required, you can ensure that the DRC errors marked as known good do not appear in the error marker spreadsheet as well as on the Layout board. To hide the known good DRC errors, follow the steps listed below.

1. From the Auto drop-down menu, choose Design Rule Check.
2. In the Check Design Rules dialog box, select the Hide Known Good DRC check box, and click OK to close the dialog box.

Important

The Hide Known Good DRC check box is enabled only if the Do not delete Known Good DRC check box is selected.

If you now view the Layout board or open the Error Markers spreadsheet, the know good DRCs will not show in the board as well as in the error list.

Note: The status of the Hide Known Good DRC check box is saved in the LAYOUT.INI file.

Modifying the color of known good DRC marker

The color of the good DRC marker can be specified in the `layout.ini` file. The steps to be followed for changing the marker color are listed below.

1. Open `layout.ini` in a text editor.
2. The RGB keyword in the [PREFERENCE] section, describes the color of the known good DRC.

   For example, RGB=(255,255,255) will set the marker color to white. Similarly, to set the marker color to orange, modify the entry to RGB=(255,155,0)
3. Restart Layout.

   The known good DRC markers on the layout board on all layers and in the Error Markers spreadsheet will appear in the color specified by you.

Delete Violating Tracks

Delete Violating Tracks removes the errors, allowing you to reroute the problem area.

To delete violating tracks

1. From the Auto menu, choose Delete Violating Tracks, then choose Board

or
from the Auto menu, choose Delete Violating Tracks, then choose DRC/Route Box.

Validating Gerber connectivity using an IPC-D-356 netlist

You can compare an IPC-356 netlist with a GerbTool netlist to detect opens and shorts that Design Rule Check and Statistics in Layout cannot detect. You create the IPC-356 netlist and a .GTD file using Layout. You create the GerbTool netlist and run the comparison in GerbTool. GerbTool creates a report that lists any discrepancies found.

Mandatory Conditions

The GerbTool IPC-356 netlist comparison will only work if the following conditions are met in Layout:

- The board must be 100% routed, with no partials routes or unroutes.
- There must be no errors reported when Route Spacing Violations, Copper Continuity Violations and Check Copper Pour are checked, using Design Rule Check.
- The Flood Planes/Pours option cannot be assigned on a Plane Layer of any padstack.
- The board cannot have any padstacks defined as blind or buried.

Creating the Gerber files in Layout

1. Save the .MAX file to a new, empty, temporary location.
2. From the Options menu, Choose Post Process Settings. The Post Process spreadsheet appears.
3. Select all etch layers (Routing and Plane) and select Properties from the pop-up menu. The Post Process Settings dialog appears.

4. Check the Enable for Post Processing option and click OK.
5. Select all of the layers that are Batch Enabled and from the pop-up menu, choose Properties.
6. In the Post Process Settings dialog box:
   a. Clear the Center on page and Mirror check boxes.
   b. Ensure the X Shift and Y Shift values are both zero. This ensures that the coordinates in the IPC-356 netlist match the coordinates in the Gerber files.
   c. In the Format group box, choose RS-274D. Do not use Extended Gerber.
   d. Click OK.
7. Left click the spreadsheet and from the pop-up menu, choose Run Batch to create the Gerber files and .GTD file. No thruhole.tap file is required.
8. Close the Post Process spreadsheet.
9. Save and close the .MAX file.

Creating the IPC-356 netlist in Layout

1. In the Layout session window, from the File menu, point to Export and choose Layout to IPC-356.
2. In the Input Layout File box, browse to the .MAX filename.
3. The file name in the Output File box will have a .NET extension. Change it to an .IPC extension. This makes it is easier to find in GerbTool.
4. In the Output format group, check the Fixed length records option. No other options need to be checked.
5. Click the Translate button to create the netlist.

Running the netlist comparison in GerbTool

1. In the Layout session window, from the Tools menu, point to GerbTool and choose Open.

2 Navigate to the .GTD file and click Open.

3 From the Setup menu, choose Layers.

4 In each layer, click the Type cell and select the correct layer type in the drop down box. Use Top, Bottom, Inner or Plane. Click OK when finished.

Note: If you changed a layer type in Layout it will come into GerbTool as the original type. You must set it to the new, correct type in GerbTool. For example: If you changed an Inner Layer to a Plane Layer in Layout, it will come into GerbTool as an Inner Layer. You must change it to Plane Layer in GerbTool.

5 Ensure that the top layer is the first layer in the list, and the bottom layer is the last layer. Use the Cut and Paste Below or Paste Above commands to change the order of the layers.

6 Left-click inside the name field of the next available layer entry below the Bottom layer. The data column should say "No" which indicates that there is no data on this layer yet.

7 Name the layer "Drill" and set the Layer type to "Drill".

8 Left-click inside the name field of the next available layer entry below the Drill layer. The data column should say "No" which indicates that there is no data on this layer yet.

9 Name the layer IPC and keep the Layer type set to "Other".

10 Click OK.

11 Select the Drill layer as the active layer from the drop-down layer list.

12 From the File menu, point to Import and choose Drill. The Import Drill dialog appears.

13 From the Files of Type drop down box, choose All Files (*.*).

14 Navigate to the THURHOLE.TAP file created by Post Processing in Layout and click Open. The Drill Format dialog box appears.

15 Accept the default values and click OK.

16 Select the Drill layer as the active layer from the drop-down layer list.

17 From the File menu, point to Import and choose IPC-D-356. Select the .IPC file in the Open dialog and click OK.

18 In the Import IPC-D-356 dialog box, uncheck both options and click OK.

19 You will see a message that indicates that no netlist is present. This message refers to the netlist that GerbTool generates to compare against the IPC-356 netlist.

20 Click Yes and GerbTool creates the new netlist.

21 Click OK in the dialog box where GerbTool indicates the number of errors found.

22 Click Yes in the dialog box where GerbTool asks if you want to view the errors.

23 A list of the errors is opened in Notepad. Use the following table to interpret the errors.

Error Interpretation

Error	Explanation
IPC Net Re-assignment, GerbTool nets 156 192 Locations: 0.7835, 0.1875 and 1.5791, 0.1850 IPC net PLLVEE:L3(1)	This is an open detected in the IPC-D-356 netlist. The items at the two locations are listed as belonging to the PLLVEE net in the IPC netlist, but GerbTool's internally generated netlist based on the gerber data show no connectivity between the items in the artwork.

Gerber Net Re-assignment: GerbTool net 2409 Locations: -0.0440, -1.5500 and 2.0050, 2.4450 IPC nets GND:~FV102(FV) DIFFSNS:P1(16)	This is a short detected in the IPC-D-356 netlist. The items listed ~FV102 and P1.16 are listed in the IPC netlist as belonging to two different nets: GND and DIFFSNS. Yet, GerbTool's internally generated netlist (based on the gerber data) has found connectivity between the two items and has assigned them to net number 2409 in GerbTool's internal database.
No IPC data for location 0.7920, 0.0562 Layer:2 No Gerber data for location 0.0569, 0.5581 ID N/C:J3(M1) idx 0	These errors usually appear if the bottom etch layer has not been moved so that it is the last etch layer listed among the gerber layers loaded into the current design.
No Gerber data for location 1.5980, 4.3800 ID 45:() idx 43	There was an IPC-D-356 record for this location, but no Gerber data.
No IPC data for location 2.8750, 3.7500 Layer:1	There is a pad on this layer that does not have any matching IPC information.

The netlist comparison performed by GerbTool is nodelist-based. If you are using surface mount technology and have the usual number of vias, GerbTool can pinpoint a short to a fairly small region. This is because the IPC-D-356 netlist describes not just the component pads, but the net for every via in your design. If your design is nothing but thru-hole technology and contains no vias at all, a short detected by GerbTool can only identify the two nets involved, but will not be able to tell you the location.

Cleaning up your design

Cleanup Design checks for aesthetic and manufacturing problems (such as off-grid 90° angles, acute angles, bad copper share, pad exits, and overlapping vias) that might have been created in the process of routing the board. You should always run Design Rule Check after running Cleanup Design.

To clean up your design

1 From the Auto menu, choose Cleanup Design.

Adding stackup data

Stackup Editor provides users a simple, intuitive method of defining and editing a stackup. It is a tool using which a user can map logical stackup of Layout to a physical stackup on a printed circuit board.

You can use Stackup Editor to,

- customize the stackup entries in a predefined board and use these to create new boards. See Customizing stackup data in a template file on page 483.
- add and modify stackup information to an existing board. See Add/modify stackup information on an existing board on page 484.

Customizing stackup data in a template file

The template files supported by Layout provide stackup information stored in them. Using the Stackup Editor, you can modify the stackup information stored in these files and save the information in a new `.tch` or `.tpl` file. You can now create new boards based on the template files.

To customize stackup settings

1 In the LSession window, from the Tools drop-down menu, choose Define Stackup.

2 In the Create Stackup dialog box, specify the path to a predefined technology template (`.tch`) or board template (`.tpl`) file to be used for creating the layout board.

3 Click OK.

Stackup Editor appears with default entries for each field in the stackup editor.

4 Modify the settings as per your requirement.

5 Click Save As.

Use of Save button is not recommended, as it will modify the predefined technology or template file you selected in Step 2 with your changes.

6 Specify a filename and click Save.

Add/modify stackup information on an existing board

1 From the Options drop-down menu, choose Stackup Settings.

Stackup Editor appears.

2 Specify the physical properties of the PCB layers.

To know about each field in the Stackup editor, see Stackup Editor dialog box on page 826.

3 In the Drawing Layer field, specify a documentation layer from the Layout drawing on which stackup information is to be added.

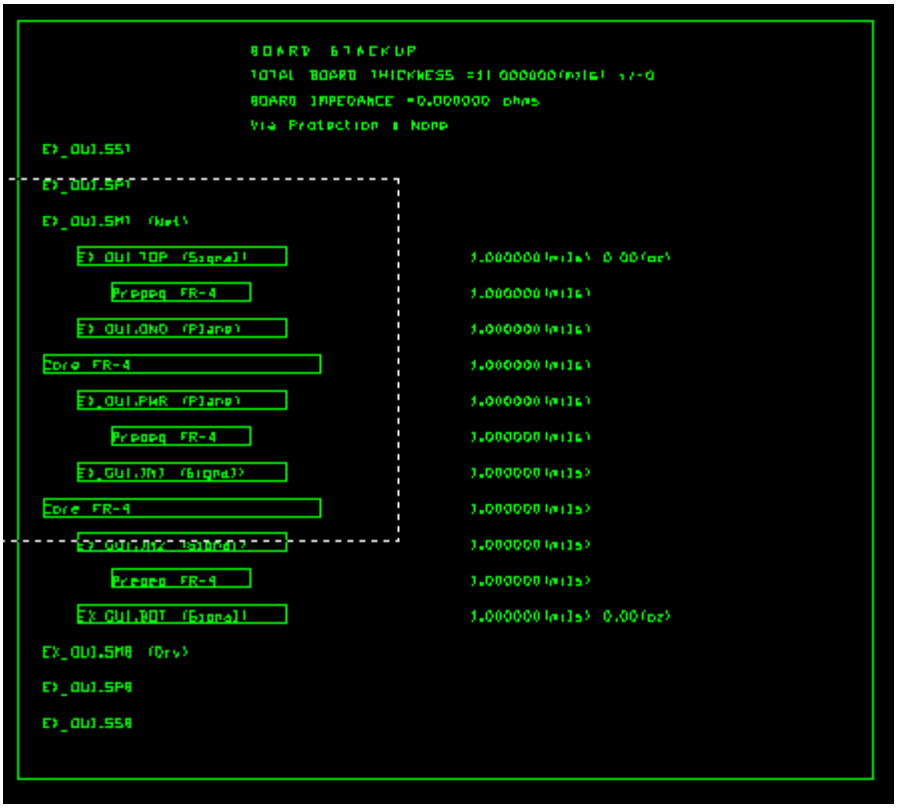
For example, selecting SMTOP as drawing layer field will add the stackup information on the SMTOP layer.

For the stackup information to display, the selected document layer must be enabled.

4 To save your settings and close the stackup, click OK.

The changes to layers made in the Stackup Editor are reflected in the board. The stackup information is displayed on

the board using a special component called a stackup component, as shown below:



The stackup component is named BOARD STACKUP.

Caution

You can use Stackup Editor to swap layers for boards designed in a release prior to 10.5. But after you have updated the board, it is recommended that the modified design files should not be used in old OrCAD releases. However if required, these files can be viewed in an older release.

Creating a new board with customized stackup settings

1 To create a new board design, you need to specify a template file to be used for creating the board layout.

Specify the name of the customized technology template file you created in the section Customizing stackup data in a template file.

2 Perform the rest of the steps for creating the board as explained in Chapter 5, “Setting up the board.”

3 To display the stackup information, launch the Stackup Editor from Layout. From the Options menu in Layout, choose Stackup Settings.

Stackup Editor appears. Notice that the values displayed in various fields is same as the values specified by you in the customized template file.

4 Specify the documentation layer on which the stackup information is to displayed.

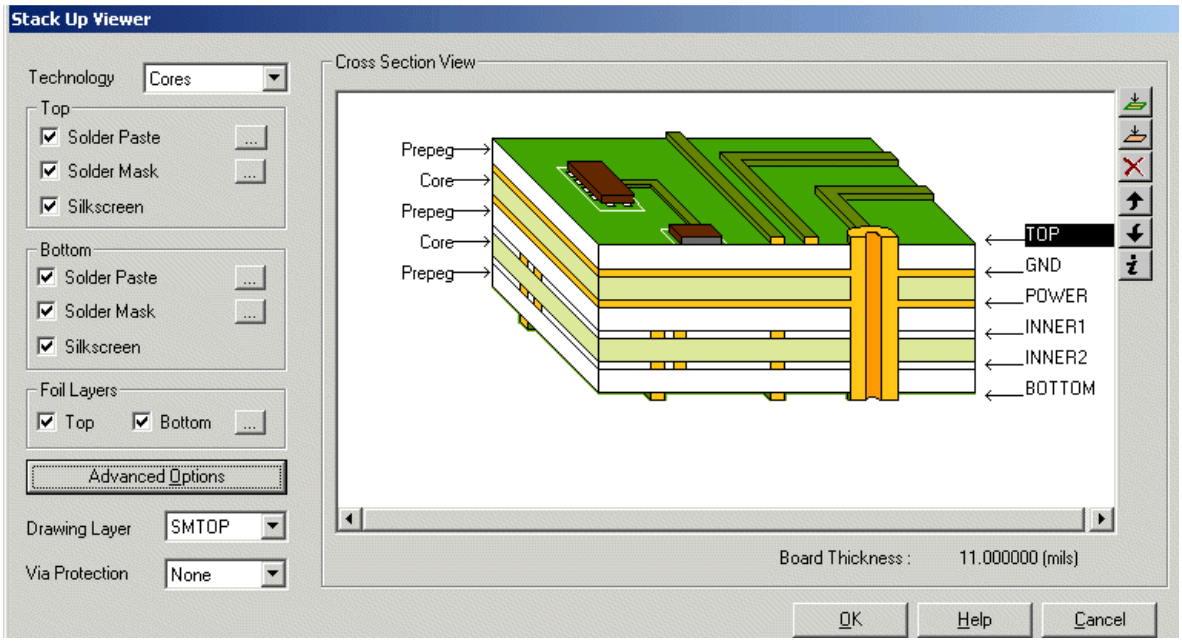
5 To display the stackup data, click OK.

Modifying board stackup data

In this section we will see, how changes in a OrCAD Layout design are reflected in a board stackup and vice-versa.

Modifying number of layers

Stackup Editor for a four layer PCB is shown in the figure given below.



The PCB has two routing layers, INNER1and INNER2, and two planes, GROUND and POWER. The layer spreadsheet for the same design is shown in the figure below.

Layers

Layer Name	Layer Hotkey	Layer NickName	Layer Type
TOP	1	TOP	Routing
BOTTOM	2	BOT	Routing
GND	3	GND	Plane
POWER	4	PWR	Plane
INNER1	5	IN1	Routing
INNER2	6	IN2	Routing
INNER3	7	IN3	Unused
INNER4	8	IN4	Unused
INNER5	9	IN5	Unused
INNER6	Ctrl + 0	IN6	Unused
INNER7	Ctrl + 1	IN7	Unused
INNER8	Ctrl + 2	IN8	Unused
INNER9	Ctrl + 3	IN9	Unused
INNER10	Ctrl + 4	I10	Unused
INNER11	Ctrl + 5	I11	Unused
INNER12	Ctrl + 6	I12	Unused
SMTOP	Ctrl + 7	SMT	Doc

We will now update the spreadsheet to add one more routing layer.

1 Select the INNER3 row and right-click.

2 From the pop-up menu, choose Properties.

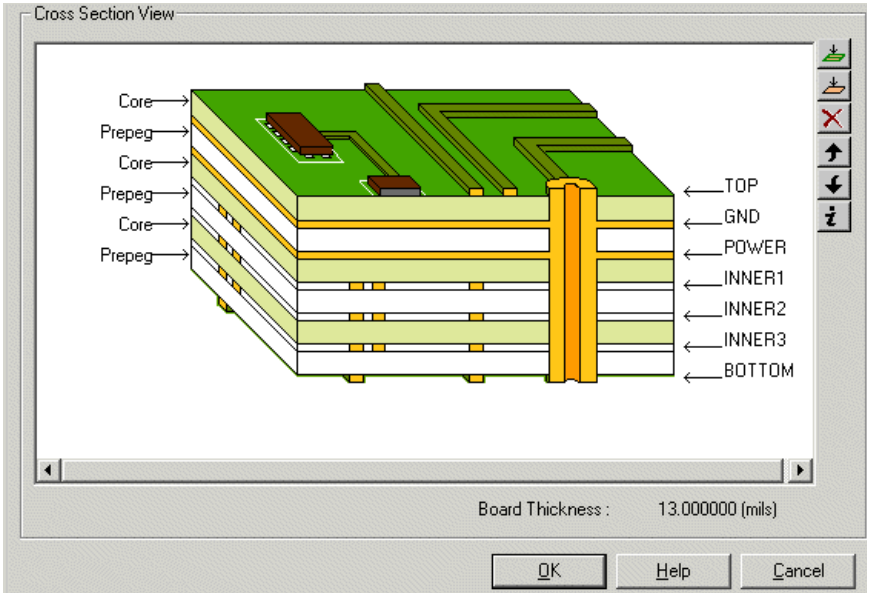
3 In the Edit Layer dialog box, set INNER3 as the routing layer. To do this, select the Routing Layer option and click OK.

The layer type for INNER3 layer changes to Routing.

If you now open the Stackup Editor, you can see the changes reflected in the Stackup as well.

1 Launch the Stackup Editor.

A new signal (routing) layer INNER3, is added to the Stackup.



2 Click OK, to close the stackup.

The stackup information displayed in the SMTOP page also reflects the changes.

Similarly, if you add or delete a layer or a plane in the stackup, the stackup information in the documentation layer gets updated immediately. For example, modify the stackup by deleting the ground plane.

a. Select the plane name GND.

b. Click the delete ( ) button.

c. Click OK to save your changes.

If you now open the Layers spreadsheet, the GND layer would be marked unused.

Using Stackup Editor, you can also change the positions of layers and planes. In such cases, information related to color, padstacks, connection, obstacles, texts, error and nets are also updated in the Layout database.

If you modify the position of layers in the board using the Stackup Editor and then load a new routing strategy file, ensure that the order of layers in the strategy file and the board are the same. If the order of layers in the strategy file and the board are not the same, the routing information in the strategy file will not be correctly loaded on the board and you will have to manually correct the strategy information.

Modifying post processing settings

Any change in the name of the output files is also reflected in the stackup information displayed on the documentation layer of OrCAD Layout. For example, if you specify a new name for the output file, this will be reflected in the displayed stackup information.

1 From the Options drop-down menu, choose Post Processing Settings.

2 Select the row with output filename as *.TOP.

3 Right-click and select Properties.

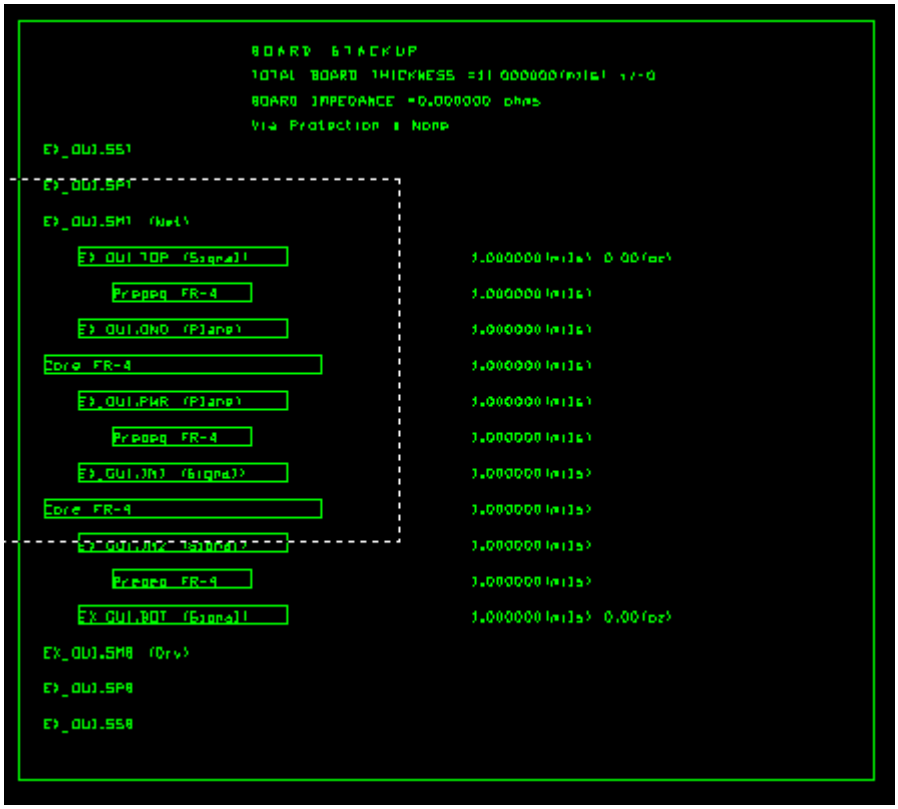
4 In the Post Process Settings dialog box, specify the filename as `sample.TOP` and click OK.

5 To refresh the displayed stackup data, press F5.

The updated stackup data is visible.

Deleting board stackup data

The stackup information is displayed on the board using a special component called a stackup component, as shown below:



The stackup component is named BOARD STACKUP.

If you do not want the stackup information to be displayed on the board, you can delete the stackup component by doing one of the following:

- Select the stackup component displayed on the board and delete it.

  To select the stackup component, press the left mouse button and drag the mouse over the stackup component.

- Open the Components Spreadsheet and delete the BOARD STACKUP component.

When you delete the stackup component, the following message appears:

```
Do you want to delete the stackup information also from the board?

Click Yes to delete the stackup information.

Click No to delete only the stackup component. You can later place the stackup information on the board. The stackup information will be passed to translators even if you do not place it on the board.
```

You can do one of the following:

- Click *Yes* to delete the stackup component and the stackup information from the board.

  Note: If you delete the stackup information only the stackup properties you added in the Stackup wizard are deleted. The changes you made to the layers in the Stackup Wizard will be retained in the board.

- Click *No* to delete only the stackup component from the board. The stackup information is retained in the board. If you later want to display the stackup information on the board, do the following:

  a. From the Options menu in Layout, choose Stackup Settings.

  The Stackup Editor appears.

  b. Click OK.

  The stackup information is displayed on the board using the stackup component named BOARD STACKUP.

Important

If you delete the stackup component but do not delete the stackup information, the stackup information is retained in the board. Even though the stackup information is not displayed on the board, it will be passed to translators such as ODB++, IPC386 and GenCAD.

Post processing

14

This chapter explains the steps you should take to finish your board. They include:

- renaming your components
- back annotating the board information to the schematic
- documenting board dimensions
- previewing the layers
- running the post processor
- creating reports

Renaming components

The Rename Components command uses the settings in the Rename Direction dialog box to rename your components in the order you specify (for example, if you choose the Up, Left strategy, Layout begins at the lower right of the board, renames components in a sweep from bottom to top, then moves to the left and renames in successive sweeps). To prevent a component from being renamed, set the Do Not Rename flag for the component before running Rename Components.

To rename components

1. From the Options menu, choose Components Renaming. The Rename Direction dialog box appears.
2. Select one of the renaming strategies, then choose the OK button.
3. Choose the spreadsheet toolbar button, then choose Components. The Components spreadsheet appears.
4. Select the components you do not want renamed, then choose Properties from the pop-up menu. The Edit Component dialog box appears.
5. Select the Do Not Rename option, choose the OK button, then close the Components spreadsheet.
6. From the Auto menu, choose Rename Components. Layout renames the components.

Annotating and cross probing

Back annotating

The Back Annotate command creates a file with a .SWP extension and puts it in the same folder your board is in. You then read the .SWP file into Capture in order to update the schematic that corresponds to your board with any changes you made to the board while it was in Layout.

Note: For information on reading a .SWP file into Capture, see the Capture documentation.

If you create a .SWP file and then run Back Annotate again, Layout prompts you to save your board file, to keep your board file synchronized with your .SWP file.

Note: You must read your .SWP file into Capture before creating another. Otherwise, the next back annotation overwrites what had been in the .SWP file. This means that your board and the schematic will become unsynchronized and you will not be able to resynchronize them.

To back annotate

1 From the Auto menu, choose Back Annotate.

Elements that do not back annotate from Layout to Capture

The following elements do not back annotate from Layout to Capture. The recommended method for performing these types of changes is to make the change in Capture and update Layout through the forward annotation process. If you choose to make the changes in Layout, be sure to manually update your schematic design.

- Changing a net. If you change net connectivity, only valid pin and gate swaps will be back annotated.
- Adding connections in Layout.
- Deleting connections in Layout.
- Adding a component to a design in Layout.
- Deleting a component in Layout.

Related topics

AutoECO

Forward annotating

Layout's AutoECO utilities are designed to automatically forward annotate all schematic attributes, component, and netlist changes to a board.

AutoECO passes information to Layout from OrCAD Capture for Windows, or from other schematic applications. The essential function of this link is to keep the schematic and board synchronized, so that any time you make a change to a schematic, it will be automatically carried forward to the board. Also, if your schematic application supports back annotation, certain changes to a board will be automatically carried backward to the schematic.

Layout, through its AutoECO utility, accepts a MAX board, or MNL, file as input, and merges it with a .TCH file to generate a board file.

For information about running AutoECO using Capture and Layout, see AutoECO.

The most common problems you are likely to have when running AutoECO or forward-annotating a schematic to a board are these:

- You did not package all of the components at the schematic level. If so, go back into your schematic design application and package the components, then run AutoECO.
- Not all of the footprints were found. If you get the message "Footprint not found," the parts were listed in the SYSTEM.PRT file, but the footprints were not found in the LAYOUT.INI library files. For information on linking footprints to components, see Link Footprint to Component dialog box.

If you make changes to your board design in Capture, you can bring those changes into Layout. A LAYOUT.INI file must exist in the \TOOLS\LAYOUT directory in order for Capture to generate a netlist or perform a forward annotation to Layout. In addition, you must save your Capture design before you can create a netlist.

Use the following procedures to forward annotate a board from Capture to Layout:

To start forward annotating schematic information from Capture

1. In Capture, open the design for which you are going to create a netlist.
2. From the Tools menu, choose Create Netlist.
3. In the Create Netlist dialog box, choose the Layout tab.
4. If the board is presently open in Layout, select the Run ECO to Layout option.

5 In the Netlist File entry box, enter a name for the output file using an .MNL file extension.

6 Choose the OK button to close the Create Netlist dialog box and create the .MNL file.

To finish forward annotating schematic information to Layout

1 If the board is presently open in Layout, and you selected the Run ECO to Layout option in Capture, make Layout the active window.

2 Choose the YES button when Layout asks if you want to update the board. Layout updates the schematic information to your board.

or

1 From the Tools menu in the session frame, choose AutoECO/Override.

2 In the File A (original design file) dialog box, select an input file with a .MAX extension, then choose the OK button.

3 In the File B (new netlist) dialog box, select the netlist file with the .MNL extension that you created in Capture, then choose the OK button.

4 In the Output report dialog box, select a name for the new output report list file with an .LIS file extension, then choose the OK button.

5 In the Merged Output Binary dialog box, select a name for a new output file with a .MAX file extension, then choose the OK button.

If Layout is unable to find a designated footprint, a dialog box allowing you to link footprints with components displays. For information on this dialog box, see Link Footprint to Component dialog box.

Related topics

Creating a new Layout project from a Capture design

Back annotating

Technology templates

AutoECO

Link Footprint to Component dialog box

Cross probing

Cross probing allows you to select one or more objects in Layout or Capture, and have the corresponding objects highlighted in the other tool. For example, with cross probing enabled, by selecting a net segment in a Capture schematic, you cause Layout to highlight the corresponding net in the board representation.

Enabling cross probing in OrCAD Capture for Windows

To use cross probing, you must have created a netlist and have the same design open in Layout as in Capture. In Capture, you must enable intertool communication (ITC).

To enable ITC in Capture

1. Choose Preferences from Capture's Options menu. Capture displays the Preferences dialog box.
2. Choose the Miscellaneous tab.
3. Activate the Enable intertool communication check box and choose the OK button.

Enabling cross probing in OrCAD Layout

It is not necessary to enable ITC in Layout, because cross probing is always active in Layout. However, you must select the Half Screen command in Layout to properly utilize ITC.

To select Half Screen in Layout

1. Launch OrCAD Capture and open your schematic.
2. Launch OrCAD Layout and open your board design.
3. In Layout, from the Window menu, choose the Half Screen command.

Cross probing from Capture to Layout

When you select certain items in your Capture schematic, cross probing highlights the corresponding components of your Layout board representation. That is, when you select a part or gate in a multiple-part package, cross probing highlights the corresponding component in the board representation. When you select a wire segment or net, cross probing highlights the corresponding net (in its entirety) on the board representation.
Any action you perform to select an object in your Capture schematic (selecting via the mouse, by using the Find command, or by performing a Browse of parts) causes the corresponding object in the board representation to be highlighted. For more information, see the following table:

Selecting this in Capture	Highlights this in Layout
Part	Corresponding component
Gate (multiple parts per package)	Corresponding component
Wire segment	Corresponding net
Net	All routes for the net
Pin on part	Corresponding pad on the component
Block selection of parts, pins, and wires	Corresponding components, pads, and nets

Cross probing from Layout to Capture

When cross probing is enabled, selecting objects in your board representation causes Capture to highlight the corresponding items in the schematic. Specifically, selecting a component (or a component pad) causes Capture to highlight all the schematic parts included in that component. Selecting a route or net causes Capture to highlight the corresponding schematic net.

Any action you perform to select an object in your Layout board representation (selecting via the mouse, from the Query window, or via the Find command) causes the corresponding object in the Capture schematic to be highlighted. For more information, see the following table:

Selecting this in Layout	Highlights this in Capture
Component	All parts in the package
Route	Corresponding wire connection
Net	Corresponding nets
Pad on component	Corresponding part
Block selection	Corresponding parts and wires

Related topics

Half Screen command (Window menu)

Retaining non-electrical parts during AutoECO

Non-electrical parts, such as mounting holes are not in the netlist, and are subject to being deleted when you run AutoECO. Because of that, you should check the Not in Netlist flag in the Edit Component dialog box before running AutoECO.

To set the Not in Netlist flag and retain non-electrical parts:

1. Choose the Spreadsheets toolbar button.
2. Select Components from the drop-down list.
3. Locate the component and double-click on it. The Edit Component dialog box displays.
4. Select the Not in Netlist check box.
5. Click OK.

Updating the footprint of a component using AutoECO

To update the footprint of a component in an existing board, it is necessary to back annotate board information to Capture, change footprint information, and then forward annotate the changes back to Layout. This ensures that the board file and schematic remain synchronized.

To update the footprint of a component in an existing board:

1. Create a swap (.SWP) file in Layout using the Back Annotate command.
2. Back annotate the swap file to Capture. For detailed instructions, see Back annotating.
3. Using the Property Editor, edit the footprint name for the desired component.
4. Save the Capture design.
5. Forward annotate schematic changes to Layout. Use the AutoECO / Override All option from the session frame. For detailed instructions see Forward annotating.

Comparing the connectivity of a Layout board file against the schematic netlist

The AutoECO utility can be used to verify that the connectivity of a Layout board file (.MAX) agrees with the connectivity contained in the schematic netlist (.MNL).

To verify that a board file has identical connectivity as the schematic, simply run AutoECO to create a new board file and verify that the new board is 100% routed.

1. Create a netlist in Capture for the schematic. For detailed instructions see Forward annotating from Capture to Layout.
2. In the Layout session window, from the Tools menu, point to ECOs, then choose AutoECO.
3. Follow the instructions for running an ECO in Forward annotating from Capture to Layout. Name the new .MAX file differently than your current board.

   Note: Pay close attention to the second AutoECO dialog box. If any ECO changes are listed that you do not expect, choose the Discard this ECO button, and check that you are using a current netlist.
4. Open the updated board file and check that the board is 100 routed. From the View menu, point to Database Spreadsheets, choose Statistics, then verify that there are no unrouted or partial connections listed in the Total column in the Statistics spreadsheet.

User defined schematic attributes

Schematic attributes that you assign to nets, parts, and pins in Capture can be transmitted to your Layout design. In Capture, simply add a user-defined property name, and assign a value to it. The property name must be uppercase as given below. The value, though assigned in Capture, is interpreted with its Layout meaning.

Schematic attributes are brought into Layout by using the AutoECO/Override Attrs command. For more information about this feature, see the AutoECO help topic.

User-defined part, or component properties

- COMPFIXED - If the value is YES, the part is permanently fixed to the board.
- COMPGROUP - An integer value that assigns the part to a group for placement.
- COMPKEY - Used to designate a component as the key component in a given group. The key component is placed first, with all the other components in the group placed in proximity to it.
- COMPLOC - Part location on the board as X and Y coordinates. Use the following format [X, Y], where X and Y represent the coordinates. Both must be integers in mils or microns.
- COMPLOCKED - If the value is YES, the part is temporarily locked in position.
- COMPROT - Part rotation in degrees and minutes counterclockwise from the orientation defined in the Layout library. Use a period (.) to separate degrees and minutes.
- COMPSIDE - Determines which side of a board a part will reside on, TOP or BOT.
- FOOTPRINT - An explicit definition of the footprint name to attach to the component.
- FPLIST - Comma-delimited list of alternate footprints to attach to components, to ease switching between footprints.
- GATEGROUP - Identifies gate swapping restrictions within a component. In order to be swapped, two gates must belong to the same gate group.
- MIRRORFOOTPRINT - An explicit mirror shape for the component, in the event you don't want Layout to perform mirroring.
- PARTNUM - A customer part number that is generally unique for each customer and identifies the exact part, including manufacturer and case type.

- PARTSHAPE - A generic part number (such as 74LS04 or CK05) that represents a certain part throughout the industry, but may not identify the manufacturer or case type. If no footprint is defined, or the correct footprint isn't found, PARTSHAPE's value is compared to the data in SYSTEM.PRT (in TOOLS/LAYOUT/DATA) and the footprint listed in SYSTEM.PRT is used.
- POWERPIN - Defines non-wired pins (such as unusual voltages) as belonging to a particular net. POWERPIN is typically used to override the standard GND or VCC attachments to particular pins of an IC.

User-defined net properties

- CONNWIDTH - Sets the track width, leaving MINWIDTH and MAXWIDTH at their defaults.
- HIGHLIGHT - If the value is YES, the net is highlighted.
- MAXWIDTH - Sets the maximum track width.
- MINWIDTH - Sets the minimum track width.
- NETGROUP - Identifies grouped nets. Use this to select or color nets as a group in Layout (for editing or routing).
- NETWEIGHT - Integer between 1 and 100 assigning relative priority to the net. The default value is 50.
- RECONNTYPE - Specifies the reconnect rules for each type of reconnect. Values are STD, HORZ, VERT, MIN, MAX, or ECL.
- SPACINGBYLAYER - Net spacing for one or more layers.
- TESTPOINT - If the value is YES, a test point is automatically assigned to the net.
- THERMALLAYERS - Comma-delimited list assigning the net to specific plane layers.
- VIAPERNET - Via types allowed for net.
- WIDTH - Track width value assigned to the MINWIDTH, MAXWIDTH, and CONNWIDTH properties unless overridden.

- WIDTHBYLAYER - Net width for one or more layers.

User-defined pin properties

- PINGROUP - Identifies pin swapping restrictions within a gate. In order to be swapped, two pins must belong to the same gate, and to the same pin group.

Documenting board dimensions

The dimension tool can create complete dimensioning objects for your board, including arrows, lines, and text. You may want to use it to show the measurements of the entire board, or to show the measurements of an object on the board, such as a large mounting hole. There are two dimension types you can choose between in the Autodimension Options dialog box: relative dimension and absolute dimension.

- Relative dimension causes a temporary origin to be created at the starting point of a drawing. The point at which you begin drawing registers as coordinates [0,0] temporarily, allowing you to easily draw the object to the dimensions you desire. The dimensions of the obstacles are measured relative to the temporary origin and the dimensioning tool draws a line and its dimension on the screen.
- With absolute dimension, the origin is fixed at the board datum. The dimensions of the object are measured from the starting coordinates as determined by the placement of the pointer relative to the board datum. The dimensioning tool only displays the coordinates at the location that you place them. It places coordinates on the X or Y axis, depending on the direction in which you begin moving the mouse.

To document board dimensions

1. From the Tool menu, choose Dimension, then New.
2. From the pop-up menu, choose Properties. The Autodimension Options dialog box appears.

3. Select Relative Dimensions.
   or
   Select Absolute Dimensions.
4. Select Open Arrow.
   or
   Select Solid Arrow.
5. In the Line Width text box, enter a value for the width of the dimension marks.
6. In the Text Height text box, enter a value for the height of dimension text.
7. From the Layer drop-down list, select the layer you want the dimension information to display on.
8. Choose the OK button.
9. In absolute dimensioning, position the cursor over the desired starting coordinates, and then click the left mouse button to begin measuring. Drag the cursor to measure, then click the left mouse button to place the first value. Repeat the process for each desired value.
   or
   In relative dimensioning, position the cursor over the desired starting coordinates, click and release the left mouse button, and move the pointer to interactively display the dimensions of the object you are measuring. Click the left mouse button again to stop measuring.

Note: Dimension uses the unit of measure you set in the Display Units group box in the System Settings dialog box (from the Options menu, choose System Settings).

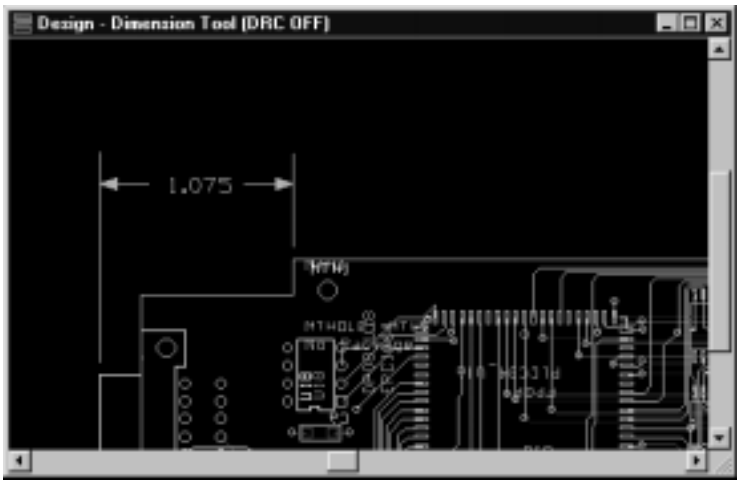


Figure 14-1 Results of using the dimension tool.

To delete dimension objects

1. From the Tool menu, choose Dimension, then Select Tool.
2. Select a dimension object.
3. From the pop-up menu, choose Delete.
   or
   Press the DELETE key.

Related topics

Select Tool command (Obstacle)

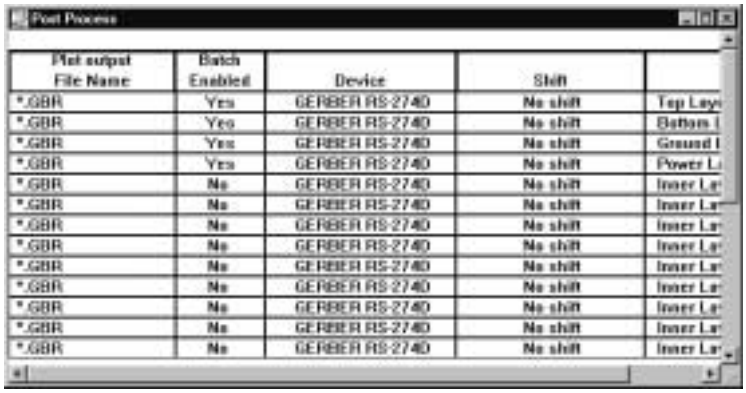
Select Tool command (Text)

Viewing the Post Process spreadsheet

In Layout, almost all post processing functions, including previewing layers, are performed using the Post Process spreadsheet.

To open the Post Process spreadsheet

1 From the Options menu, choose Post Process Settings. The Post Process spreadsheet appears.



In the Post Process spreadsheet, you can view the following information.

Plot Output File Name

Indicates the filename extensions given to the plot output files. To change file extensions, double-click in the heading cell to select all the rows in the spreadsheet and bring up the Post Process Settings dialog box. Type an asterisk and a period (*.) in the File Name text box, then choose the OK button.

Batch Enabled

Indicates whether output will be generated for the layer (“Yes”) or not (“No”). To toggle the setting, double-click in a Batch Enabled cell to bring up the Post Process Settings dialog box, select or deselect the Enable for Post Processing option, then choose the OK button.

Device

Lists the name of the target device. Layout supports either direct plotting or output to file for Gerber, Extended Gerber, DXF, and the print manager.

Note: In addition to using the print manager to specify drivers, you can choose Print from the File menu to specify standard Windows drivers, for support of devices such as PostScript or color printers.

Shift

Lists any special shifting, rotation, mirror, or scaling requirements.

Plot Title

A entry you supply that identifies generated reports and provides notes for future Layout sessions. Comments can include up to 100 characters.

Previewing layers

As you create your board, you generate the necessary artwork and labels for each layer. Before you implement post processing, you should preview each layer to ensure that all of the necessary elements are present and visible on the film that you are sending to the manufacturer.

If an item is visible on the screen in preview mode, it appears in the Gerber or DXF output. If the item is invisible on the screen, it does not appear in the output. You can preview the board layer by layer and toggle the visibility of items on the board.

Copper layers

- Verify the position of associated labels
- Check that the rotation, shift, and output format are properly set

Power planes

- Verify that thermal reliefs are present on the proper planes for the proper nets
- Ensure that the plane has proper clearance from the board edge
- Verify the position of associated labels
- Check that the rotation, shift, and output format are properly set

Silkscreen layers

- Verify the position of the reference designators
- Verify the position of other labels
- Check that the rotation, shift, and output format are properly set

Solder mask layers

- Verify the position of associated labels
- Check that the rotation, shift, and output format are properly set

Assembly drawing layers

- Verify the position of the reference designators
- Verify the position of other labels
- Check that the rotation, shift, and output format are properly set

Solder paste layers

- Verify that the proper pads are displayed
- Verify the position of associated labels

- Check that the rotation, shift, and output format are properly set

Drill drawing layers

- Verify the position of associated labels
- Review drill chart
- Move or resize the drill chart, if necessary

  Note: For information on moving and resizing the drill chart, see *Moving the drill chart* on page 516.

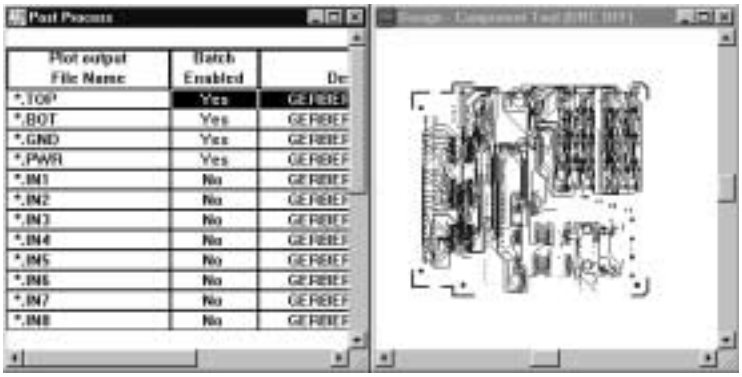
- Check that the rotation, shift, and output format are properly set

Previewing a layer

To preview a layer

1. From the Options menu, choose Post Process Settings. The Post Process spreadsheet appears.
2. From the Window menu, choose Tile so that you can view both the Post Process spreadsheet and the design window.
3. In the Post Process spreadsheet, select the layer you want to preview by clicking in the Plot Output File Name cell for the layer.

4 From the pop-up menu, choose Preview. The preview of the layer appears in the design window.



5 Check the layer preview for the items that should be visible for output. If all necessary items are visible on the layer preview, skip to step 11.

6 If an item that should be visible on the preview for a layer is not visible, choose the color toolbar button. The Color spreadsheet appears.

Note: Diagonal lines in the Color spreadsheet indicate that the object or layer is currently defined as invisible. To make items visible or invisible for preview and output, you must access the Color spreadsheet while the Post Process spreadsheet is active. When the Post Process spreadsheet is active, the visibility settings apply only to what you see in the preview window, and consequently in your output; the selections do not affect the graphical display of your board in the design window.

7 Select the item that you want to make visible, then choose Visible<>Invisible from the pop-up menu.

Note: If the item that you want to select is not listed in the Color spreadsheet, choose New from the Color spreadsheet's pop-up menu. In the Add Color Rule dialog box, select the item that you want to add, indicate the layer that you want it to display from, and choose the OK button.

8 Close the Color spreadsheet.

9 In the Post Process spreadsheet, choose Save Colors from the pop-up menu to save this setting, then choose

Preview from the pop-up menu to redraw the screen. The item should now be visible in the layer preview.

10 Repeat steps 7, 8, and 9 for each item that is invisible, but should be visible.

Note: Because the Visible<>Invisible command is a toggling command, you can also make visible items invisible using steps 7, 8, and 9.

11 Repeat this process for each layer in the Post Process spreadsheet.

12 When you're finished previewing the layers, choose Reset All from the Window menu. Layout ends preview mode, minimizes the Post Process spreadsheet, and returns the design window to its previous size.

Viewing the drill drawing and drill chart

If you wish, you can print the drill drawing layer, with the drill symbols and the drill chart.

To view drill hole positions and the drill chart

1 In the left column of the Color spreadsheet, check for the presence of Default DRLDWG.

2 If Default DRLDWG is present, proceed to step 4, below.

or

If it is not present, choose the New command from the pop-up menu to display the Add Color Rule dialog box.

3 In the Layer text box, remove the minus sign and enter DRLDWG, then choose the OK button. The Color spreadsheet is visible again.

4 Double-click on Default DRLDWG to display the Color dialog box.

5 Select a distinctive color, and choose the OK button

6 From the pop-up menu, verify that the Invisible command is not selected.

7 Close the Color spreadsheet window.

8 From the Layer drop-down list box on the toolbar, choose the Drill Drawing layer (DRD). The drill symbols display in the selected color.

9 From the Tool menu, choose the Move Drill Chart command.

10 Position the pointer where you want the upper left corner of the drill chart, and click the left mouse button. The drill chart is also visible.

Related topics

Drills spreadsheet

Restoring the design window view

To restore your design window view

1 Close the Post Processing spreadsheet.

or

1 With the spreadsheet window active, display the pop-up menu and choose the Restore command.

Note: You may be able to save time by applying one color scheme to several layers. When you have one layer set up, select the next layer in the Post Process spreadsheet and choose Preview from the pop-up menu. If everything is as you want it, update the colors and go on to the next layer.

Related topics

Gerber plot preview

Gerber plot preview

In Layout, you can check your Gerber output before you plot it.

To preview your Gerber output

1. From the Options menu, choose Post Process Settings. The Post Process spreadsheet C0363 appears.
2. From the Windows menu, choose the Tile command to display the spreadsheet and the design window side by side.
3. Activate the spreadsheet, select the layer you want to preview, display the pop-up menu and choose the Preview command. The Gerber preview displays in the design window.

Note: Customers have reported that according to the Product Comparison Guide for HP-GL/2 and HP RTL Peripherals published by Hewlett-Packard, the following devices are obsolete and no longer supported. Therefore, you may be unable to acquire a current, working driver for the following output devices.

- HP DesignJet plotters (except 200, 600, and 650C)
- HP PaintWriter XL printer
- HP DraftMaster Series plotters - SX, RX, and MX (except the SXplus, RXplus, and Mxplus)
- HP 7600 Series plotters - Models 240D/E, 250, 255, and 355

Certain companies (for example, WinLINE) produce drivers. You can contact WinLINE on the internet at http://www.winline.com.

Related topics

Gerber Preferences dialog box

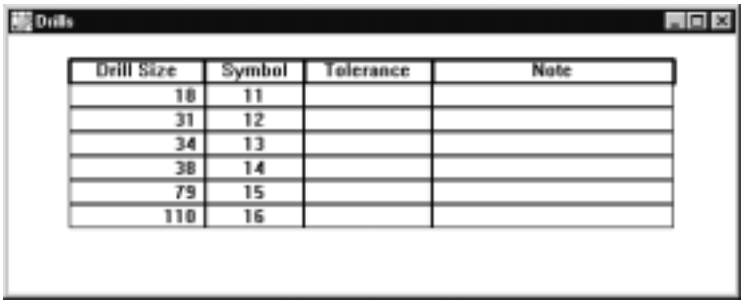
Print/Plot dialog box

Moving the drill chart

The drill chart is automatically generated, and includes the current counts of all of the existing drill sizes on the board. The drill chart comes with 20 graphical symbols (11-20 are smaller representations of 1-10) and 26 scalable alpha characters. A drill symbol is assigned to each drill size found. The symbols used for each drill and the text inside the drill chart are defined in the Drills spreadsheet. You can manipulate the size of the drill chart and move it to a location that is suitable for your board.

To view the Drills spreadsheet

1 Choose the spreadsheet toolbar button, then choose Drills. The Drills spreadsheet appears.



To change the size of the drill chart

1 Close the Drills spreadsheet if it is open.

2 From the Tool menu, choose Drill Chart, then Drill Chart Properties. The Drill Chart Properties dialog box appears.

3 Enter values for text height and line width, then choose the OK button. Layout redraws the drill chart using the new values.

To move the drill chart

1 Close the Drills spreadsheet if it is open.

2 From the Tool menu, choose Drill Chart, then Move Drill Chart.

3 Click on the new location. Layout moves the drill chart to the new location.

4 Press ESC to exit move mode.

Note: If the drill chart is not visible, choose the color toolbar button and change the color of the DRLDWG layer in the Color spreadsheet to a color that contrasts with your background color.

Generating a drill tape

When you select the Create Drill Files option in the Post Process Settings dialog box, Layout produces drill tape files (.TAP) and (.NPT) in Excellon format and places them in your working directory. During the manufacturing process, the drilling machine reads these files to determine the size and location of the drill holes on your board. Unless you shift the output using the X Shift and Y Shift settings in the Post Process Settings dialog box, the drill tape coordinates match the coordinates that you see in the design window.

Layout outputs a file named THRUHOLE.TAP for plated through-holes and a file named THRUHOLE.NPT for non-plated through-holes in your board.

Note: If you want Layout to output a single drill tape file named THRUHOLE.TAP for both plated and non-plated through holes, select the Combine Plated/Non-Plated Thru Holes check box in the Post Process Settings dialog box.

In addition, Layout automatically generates drill tape files for each layer pair that shares a blind or buried via and names them accordingly. For example, a file with the name 1_4.TAP includes data related to layers 1, 4, and all layers in between.

Note: If you want to preserve drill tape files, rename them, to avoid having them replaced with newly generated files.

To generate a drill tape

Note: Ensure that at least one of the layers has a “Yes” in the Batch Enabled column (otherwise, drill tapes are not generated). To batch enable a layer, double-click in its Batch Enabled cell to bring up the Post Process Settings dialog box, select the Enable for Post Processing option, then choose the OK button.

1. From the Options menu, choose Post Process Settings. The Post Process spreadsheet appears.
2. Click in one of the spreadsheet’s rows, then choose Properties from the pop-up menu. The Post Process Settings dialog box appears.
3. Select the Create Drill Files option.

   When you run the post processor, Layout outputs a drill file named THRUHOLE.TAP for plated through-holes and a file named THRUHOLE.NPT for non-plated through-holes.

   Note: Select the Combine Plated/Non-Plated Thru Holes option if you want Layout to output a single drill tape file named THRUHOLE.TAP for both plated and non-plated through holes.
4. Click the OK button.
5. From the Auto menu, choose Run Post Processor.
6. Respond to Layout’s notification messages that it has created a Gerber aperture file (.APP) (if you’re creating Gerber RS-274D output), a Gerber design file (.GTD), and THRUHOLE.TAP (if your board has through-holes) and THRUHOLE.NPT (if your board has non-plated through-holes).
7. Close the post processor report (.LIS) after you’ve viewed it.
8. If you want to view the drill files, choose Text Editor from the File menu, choose Open from the text editor’s File menu, change Files of type to All Files, locate the THRUHOLE.TAP or THRUHOLE.NPT file, and double-click on it.

9 Close the drill file after you've viewed it.

Using Run Post Processor

The Run Post Processor command creates files for the layers that are batch enabled in the Post Process spreadsheet. Output files are created for each layer and given appropriate file extensions corresponding to the type of output.

Note: If your output format is either Gerber RS-274D or Extended Gerber, an additional file (design_name.GTD) is created, which is a special design file preconfigured for GerbTool.

To perform post processing

1 From the Options menu, choose Post Process Settings. The Post Process spreadsheet appears.

2 Select a layer (or layers) you want to change settings for, then choose Properties from the pop-up menu. The Post Process Settings dialog box appears.

3 Select an output format, select the appropriate options (choose the dialog box's Help button for information on the options in the dialog box), choose the OK button, then close the Post Process spreadsheet.

4 If necessary, choose Gerber Settings from the Options menu. Select the options you want in the Gerber Preferences dialog box, then choose the OK button.

Note: In the User Preferences dialog box, ensure that the Enable Copper Pour option is selected before you create a Gerber plot. Otherwise, your Gerber plots will have no copper pour in them.

5 From the Auto menu, choose Run Post Processor. Layout creates the post processing files.

Creating reports

The Create Reports command brings up the Generate Reports dialog box, within which you select the output reports you would like to have generated.

To create reports

1. From the Auto menu, choose Create Reports. The Generate Reports dialog box appears.
2. Select the reports you want generated (choose the dialog box's Help button for information on the reports), then choose the OK button.

Component list and netlist report definition

Layout's two user-definable reports are Component List (COMPLIST) and Netlist (NETLIST).

A COMPLIST report is a list of components in alphanumeric order, one per line. The rest of the line can contain information pertaining to each component using the format described below.

A NETLIST report is a list of nets in alphanumeric order, one per line. The rest of the line can contain information pertaining to the net using the format described below.

You define a report format in the LAYOUT.INI file, which is located in the WINDOWS directory. Open the file in an ASCII text editor and search for the line that contains [COMPLIST] or [NETLIST] (with the brackets).

The report description can be several lines, one for each field or attribute. The report description includes the following information types:

- **FIELD** or **ATTR -** Defines the field type.
- **Width -** The next entry into the report definition is the field width in characters. Unless justification is set to one of the

wrap options, data appearing in this column will be truncated to this width.

- **Justification -** L, R, C (Left, Right, or Center) or LW, RW, CW (Left wrapped, Right wrapped, or Center wrapped) are supported as field justifications. If a field is wrapped, the text on each continuing line will end with " >" and the text will jump down to the next line within the current field width to finish the string.
- **Delimiter -** Single character (between single quotes) that is used as a delimiter between fields. The default is a space. The delimiter displays after the specified field in the report.
- **Title -** The title is user-definable, and displays at the top of each column.

Two types of information can be printed in a COMPLIST or NETLIST report; these are FIELD or ATTR.

A FIELD is definable in Layout, and is independent of the schematic input (although it may have been originally defined there).

An ATTR is a schematic attribute or property, and cannot be redefined in Layout.

To define a report format, choose from the fields and attributes appropriate to the report, and copy those lines to the LAYOUT.INI file below the line [COMPLIST] or [NETLIST], as appropriate.

The following is the list of COMPLIST fields created by Layout:

Fieldname	Width	Just.	Delim.	Title
FIELD=COMPNAME	10	L	' '	"Compname"
FIELD=X	12	L	' '	"X Coord"
FIELD=Y	12	L	' '	"Y Coord"
FIELD=XINSERT	12	L	' '	"X Insert"
FIELD=YINSERT	12	L	' '	"Y Insert"

FIELD=ROT	6	L	' '	"Rotation"
FIELD=PARTNAME	16	L	' '	"Partname"
FIELD=PACKAGE	16	L	' '	"Package"
FIELD=COMPSIDE	16	L	' '	"Compside"
FIELD=UID	6	L	' '	"Uid"
FIELD=FOOTPRINT	16	L	' '	"Footprint"
FIELD=GROUP	6	L	' '	"Group"
FIELD=CLUSTER	8	L	' '	"Cluster"
FIELD=INSERTORDER	12	L	' '	"Insertion"
FIELD=UNUSEDPINS	60	L	' '	"Unused Pins"

Fieldnames are defined as follows (these keywords are predefined):

- **COMPNAME -** Component name from Layout.
- **X -** X location of component origin from board datum.
- **Y -** Y location of component origin from board datum.
- **XINSERT -** X location of component insertion origin from board datum.
- **YINSERT -** Y location of component insertion origin from board datum.
- **ROT -** Rotation of component in degrees and minutes from its orientation in the footprint library.
- **PARTNAME -** Component part name, derived from the PARTTYPE attribute of the schematic, or manually derived in Layout using the "Component name" field (specified in the Edit Component dialog box).
- **PACKAGE -** Derived from the schematic. Usually the name of the schematic symbol.
- **COMPSIDE -** Calculated by Layout based on the board side of the place outline.
- **UID -** "Unique ID," calculated by Layout as an unchanging unique identifier, even if the component gets renamed.

- **FOOTPRINT -** Footprint used by the component.
- **GROUP -** Placement group, derived from the schematic or assigned in the Edit Component dialog box.
- **CLUSTER -** A component group defined in Layout, rather than in the schematic.
- **INSERTORDER -** Insertion order of components, if specified, to be used in pick-and-place board manufacturing.
- **UNUSEDPINS -** Component pins that are currently unused.

An ATTR (attribute) can be any property that displays in a schematic.

The following is a list of Capture properties available in a COMPLIST report:

Fieldname	Width	Just.	Delim.	Title
ATTR=FPLIST	30	L	' '	"Footprint List"
ATTR=COMPLOC	12	L	' '	"Location"
ATTR=COMPROT	8	L	' '	"Rotation"
ATTR=COMPGROUP	4	L	' '	"Group"
ATTR=COMPKEY	5	L	' '	"Priority"
ATTR=COMPFIXED	4	L	' '	"Fixed"
ATTR=COMPLOCKED	4	L	' '	"Locked"
ATTR=COMPSIDE	16	L	' '	"Compside"

The following are examples of NETLIST fields in Layout:

Fieldname	Width	Just.	Delim.	Title
FIELD=NETNAME	10	L	' '	"Net Name"
FIELD=NETPINS	60	L	' '	"Pin List"
FIELD=ROUTEDLEN	16	L	' '	"Routed Length"

FIELD=UNROUTEDLEN	16	L	' '	"Unrouted Length"
FIELD=VIAS	6	L	' '	"Vias"
FIELD=CONNECTIONS	11	L	' '	"Conn List"
FIELD=PERCENTAGE	10	L	' '	"% of total"
FIELD=COMPPINS	60	L	' '	"Component Pins"

Related topics

Create Reports command

Generate Reports dialog box

Printing and plotting

Using the Print/Plot dialog box, you can send a graphic image of your board to a printer or plotter, or to a print file.

To print an image of a board

1. Open the board in the design window.
2. Zoom in or zoom out to view the desired area to print.
3. From the File menu, choose Print/Plot. The Print/Plot dialog box appears.
4. If you want to print just the area of the board visible in the design window, select the Print/Plot Current View option.
   or
   If you want to print the entire board, set the options as desired for keeping drill holes open, centering or shifting the image, and mirroring, scaling, or rotating the image.
5. Choose the OK button. A Print dialog box for your system's printer or plotter appears.
6. Select a printer or plotter, choose the appropriate settings, then choose the OK button. The image is sent to your printer or plotter.

To send an image to a file

1. Follow steps 1 through 4 in *To print an image of a board* above.
2. Select the Print/Plot To File option, and supply a filename with a .PRN extension in the File Name text box.
   or
   Select the DXF option (which automatically selects the Print/Plot To File option), and supply a filename with a .DXF extension in the File Name text box. You cannot use the Print/Plot Current View option with the DXF option.
3. Choose the OK button. A Print dialog box for your system's printer or plotter appears.
4. Select a printer or plotter, choose the appropriate settings, then choose the OK button. The print file is created and put into your working directory.

Managing libraries and footprints

15

You can use the library manager to access and view every library and footprint supplied by Layout. You can make libraries available for the current Layout session, and can remove them from the session. You can also create custom libraries, copy footprints between libraries, and delete footprints from libraries.

This chapter explains how to manage Layout's footprint libraries and describes the following tasks.

- Opening the library manager
- Making libraries available for the current session
- Removing libraries from the current session
- Creating a custom library
- Adding and copying footprints to libraries
- Removing footprints from libraries

Libraries

Libraries are files that contain reusable board data. Layout provides the capability to develop a footprint library for component footprints. Libraries may also contain a variety of symbols that you can reuse in your boards.

The relationship between the library, and the footprints and symbols it contains, is similar to the relationship between a board and its contents. The contents of the library move with the library and are deleted with the library.

You can create custom libraries to store any combination of items. You can, for example, create a library to hold functionally related components, or to hold symbols such as alignment targets. Or, you can create a library to contain all of the footprints used in a project.

Note: You can add a library from a previous version of Layout, or add an existing board file as a library, by choosing Old Library (*.LIB) or Board (*.MAX) in the List files of type drop-down list in the Add Library dialog box (accessed by choosing the Add button in the library manager). You can't add a .MAX file you have open in Layout to its own library (you'll receive the message "The library is already loaded in the system").

Note: If you edit a library provided by Layout, you should give it a new and unique name so that it will not be replaced when you install updated libraries.

When you work with footprint libraries in Layout, you use the library manager and the footprint editor. The library manager lists the libraries and all of the footprints contained in the libraries, and the footprint editor is a graphical editing environment. You have the option of selecting libraries and footprints for editing.

Because a library is a file, you can use the same Windows principles that apply to other files when working with libraries.

Footprints

Footprints describe the physical description of components. A footprint generally consists of three object types: padstacks, obstacles (representing among other things, the physical outline of the component, silkscreen outline, assembly outline, and placement and insertion outlines), and text (for example, the component name or component value).

You can view footprint data graphically in the footprint editor or textually in the Footprints spreadsheets.

The library manager

You can start the library manager from the session frame before you open a design, or from the toolbar in the Layout design window.

Libraries may exist in any directory, even on a network. Layout allows you to use libraries from any of these sources at the same time. Although Layout ships with a set of libraries containing over 3000 parts that are installed automatically and are accessible for use, you may add additional libraries from another area.

In the Libraries window, select a library to generate and display a list of its parts in the Footprints window. If you select multiple libraries using the CTRL key, the Footprints window displays a list of the footprints in all selected libraries in alphabetical order.

When you select a footprint from the list in the Footprints window, a graphical display of the footprints appears in the footprint editor. You can perform various actions on the footprint, such as editing, saving, copying, and deleting.

Using the library manager, you can create a new library by saving a new or existing footprint to a library that you name. You can then add other footprints by selecting them in the Footprints window and saving them to the newly created library.

Using the library manager, you can add or copy a footprint to a library by saving the footprint to the desired library. You can also delete footprints from libraries.

Note: Running multiple copies of Layout is not recommended. When multiple copies are running, data is shared between the copies. Performing procedures such as AutoECO or making edits in the library manager can cause changes in other open copies.

Starting the library manager

To start the library manager

1 Choose the library manager toolbar button.
or
From the File menu, choose Library Manager.

To close the library manager, click the X in the upper, right-hand corner in either the library manager window or the footprint editor, and choose the OK button when Layout asks if you want to close the library manager. You can also close the Library Manager by clicking the Library Manager button in the tool bar.

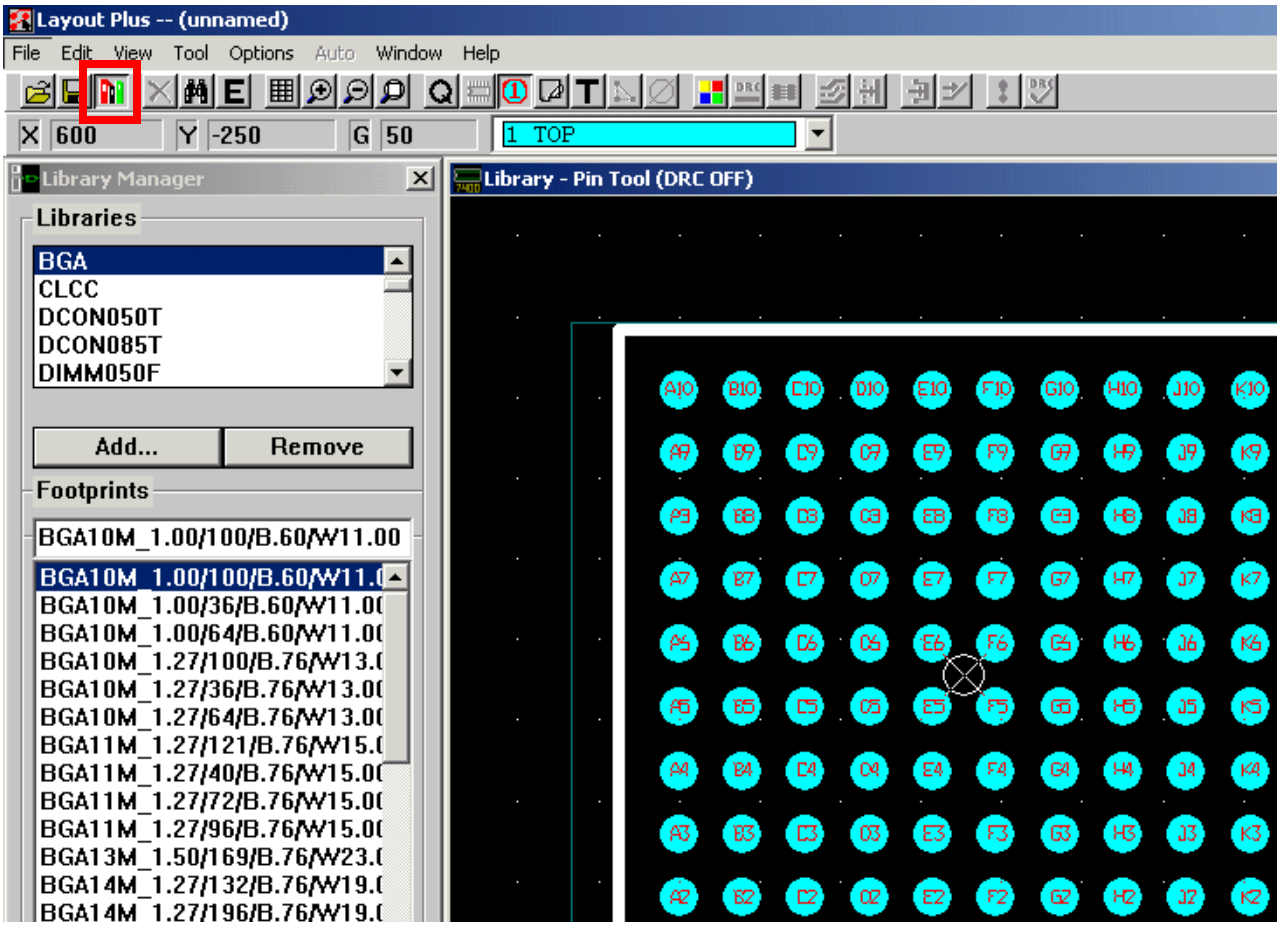


Figure 15-1 The library manager and footprint editor.

Making libraries available for use

Libraries may exist in any directory, even on a network. You can use libraries from any of these sources at the same time. Although Layout ships with a set of libraries that are installed automatically and are accessible for use, you can add additional libraries.

To make a library available for use in Layout, you use the Add Library button in the library manager. You then have access to all of the footprints in the added library.

Note: You can add a library from a previous version of Layout, or add an existing board file as a library, by choosing Old Library (*.LIB) or Board (*.MAX) in the List files of type drop-down list in the Add Library dialog box (accessed by choosing the Add button in the library manager). You can't add a .MAX file you have open in Layout to its own library (you'll receive the message "The library is already loaded in the system").

You can also remove libraries from the list of available libraries. When libraries are removed, they are not deleted. They are just removed from the list of libraries displayed in the library manager.

To make a library available for use

1. Choose the library manager toolbar button. The library manager appears.
2. Choose the Add button. The Add Library dialog box appears.
3. Locate and select the library (.LLB) that you want in the Library folder. You can select multiple libraries using the CTRL key.
4. Choose the OK button. The library is added at the top of the Libraries list.

To make a library unavailable for use

1. Choose the library manager toolbar button. The library manager appears.
2. Select a library in the Libraries list. You can select multiple libraries using the CTRL key.
3. Choose the Remove button. Layout asks you to confirm your decision.
4. Choose the Yes button. The library is removed from the Libraries list.

Viewing footprints

In the Libraries list, select a library to generate and display a list of its parts in the Footprints list. If you select multiple libraries using the CTRL key, the Footprints list displays a list of the footprints in all selected libraries in alphabetical order.

When you select a footprint from the Footprints list, a graphical display of the footprint appears in the footprint editor. You can perform various actions on the footprint, such as editing, saving, copying, and deleting it.

To view footprints in the footprint editor

1. Choose the library manager toolbar button. The library manager appears.
2. Select a library in the Libraries list. You can select multiple libraries using the CTRL key. The footprints from the selected library display in the Footprints list.
3. Select a footprint in the Footprints list. The footprint appears in the footprint editor.

Searching Footprints

Layout lets you quickly search footprints by name, minimum pin count, and maximum pin count.

When you find a footprint that meets your requirement, you can view the footprint in the footprint editor. If the desired footprint is in a library that you have not made available for use in Layout, you can make the library available for use from the Search Footprint dialog box and view the footprint. For more information on making a library available for use in Layout, see Making libraries available for use on page 531.

To search footprints in the library manager

1 Click the library manager toolbar button.

The library manager appears.

2 Click the Search button.

The Search Footprint dialog box appears.

3 Select a check box and specify the search criteria.

Select	To
Name	Search footprints by name. You can use the asterisk (*) character to perform a wildcard search. Select the Match whole word check box if you want to find only the footprints that match the footprint name you specified. For example, if you enter the footprint name as VRES1 and do not select this check box, footprints with names such as VRES1, VRES10, VRES11, and so on will be found. If you select this check box, the search will only find the VRES1 footprint.
Min. pin count	Find footprints that have at least the specified number of pins and also have the same pin names as the pin names on the part in your Capture schematic.

Select	To
Max. pin count	Find footprints that do not have more than the specified number of pins and also have the same pin names as the pin names on the part in your Capture schematic.

4 Select the Configured libraries check box if you want to search the libraries you have made available for use in Layout. For more information on making a library available for use in Layout, see Making libraries available for use on page 531.

5 Select the Other libraries check box if you want to search the libraries you have not made available for use in Layout.

a. Click the browse button to add other libraries.

The Open dialog box appears.

b. Select a library.

To select multiple libraries use the Shift or CTRL key.

Note: You cannot select libraries existing in multiple directories. Copy all your custom libraries to a single directory.

c. Click the Open button to add the library.

6 Click the Search button.

The footprints that match the search criteria and the libraries in which the footprints exist are displayed.

7 Select a library in the Libraries list to display the footprints in the library that match the search criteria.

8 Select a footprint and click the Add button to view the footprint in the footprint editor.

Note the following:

- If the library in which the selected footprint exists is not already made available for use in Layout, the

library is automatically added to the list of libraries available for use in Layout. For more information on making a library available for use in Layout, see Making libraries available for use on page 531.

- ❑ If the library in which the selected footprint exists is already made available for use in Layout, Layout displays an error message that the library is already bound to the session. Click OK to view the footprint in the footprint editor.

Creating a custom footprint library

Using the library manager, you can create a custom library by saving a new or existing footprint to a library that you name. You can then add other footprints by selecting them in the Footprints list and saving them to the newly created library.

Note: You can add a library from a previous version of Layout, or add an existing board file as a library, by choosing Old Library (*.LIB) or Board (*.MAX) in the List files of type drop-down list in the Add Library dialog box (accessed by choosing the Add button in the library manager). You can't add a .MAX file you have open in Layout to its own library (you'll receive the message "The library is already loaded in the system").

To create a custom footprint library

1. Choose the library manager toolbar button. The library manager appears.
2. In the Footprints list, select a footprint to save to the new library. The footprint appears in the footprint editor.
   or
   Create a footprint as described in *Creating and editing footprints*.

3 Choose the Save As button. The Save Footprint As dialog box appears.

4 Choose the Create New Library button. The Create New Library dialog box appears.

5 Enter the name for the new library (using a .LLB extension) in the File name text box, select a directory for the library, then choose the Save button.

6 Choose the OK button to close the Save Footprint As dialog box. The new library is added at the top of the Libraries list.

7 Add footprints to the new library by following the instructions in *Adding, copying, and deleting footprints* in this chapter.

Adding, copying, and deleting footprints

Using the library manager, you can add or copy a footprint to a library by saving the footprint to the desired library. You can also delete footprints from libraries.

To add or copy footprints to libraries

1 In the library manager, select the footprint name in the Footprints list. The footprint appears in the footprint editor.

2 Choose the Save As button. The Save Footprint As dialog box appears.

3 Select a library from the drop-down list.
or

Choose the Browse button. Locate and select the desired library.

4 Choose the OK button.

To delete footprints from libraries

1 In the library manager, select the footprint name in the Footprints list. The footprint appears in the footprint editor.

2 Choose the Delete Footprint button. Layout asks you to confirm your decision to delete the footprint.

3 Choose the Yes button. The footprint is deleted from the library.

Note: The footprint is permanently removed from the library. If there is a possibility that you will want to use the footprint in the future, you should first copy the footprint to another library, such as OLD.LLB, before you delete it.

Creating and editing footprints

A footprint is the physical description of a component and consists of three elements: padstacks, obstacles (silkscreens, assembly drawing data, outlines), and text. You can create and edit footprints in the footprint editor. You can also access and edit footprint data for the board using the Footprints spreadsheet.

Note: Running multiple copies of Layout is not recommended. When multiple copies are running, data is shared between the copies. Performing procedures such as AutoECO or making edits in the library manager can cause changes in other open copies.

Setting a grid for the footprint pins

If you are not using the Pad Array Generator, it is important to set a placement grid when creating footprints. When you start creating a new footprint, the first padstack is automatically placed at [0,0]. When you add new padstacks, they are placed

according to the placement grid specified in the System Settings dialog box (from the Options menu, choose System Settings).

For information on setting a placement grid, see *Setting system grids* on page 234.

Creating a footprint

You can create new footprints and add them to the libraries of your choice.

To create a footprint

1. Ensure that the TOP layer is selected in the Layer drop-down list box on the toolbar.

   Cadence recommends that you create footprints only on the TOP layer.

2. In the library manager, click the Create New Footprint button. The Create New Footprint dialog box appears.

3. Enter a name for the new footprint.

4. If the footprint is to be a metric footprint, select the Metric option.

5. If you would like to use the Pad Array Generator to generate and place a number of pads, select the Use Pad Array Generator option.

6. Click the OK button. If you have chosen to use it, the Pad Array Generator will launch. Otherwise, the footprint

origin, one pin, and default text objects display in the footprint editor.

Figure 15-2 The footprint editor.

Using the Pad Array Generator

The Pad Array Generator can be a useful part of creating complex footprints. If you have a large number of pads that need to be precisely placed and numbered, the Pad Array Generator can significantly decrease your footprint creation time.

To use the Pad Array Generator

1 From the Create New Footprint dialog box, select the Use Pad Array Generator option and click OK. The Pad Array Generator dialog box and Array Preview window appear.

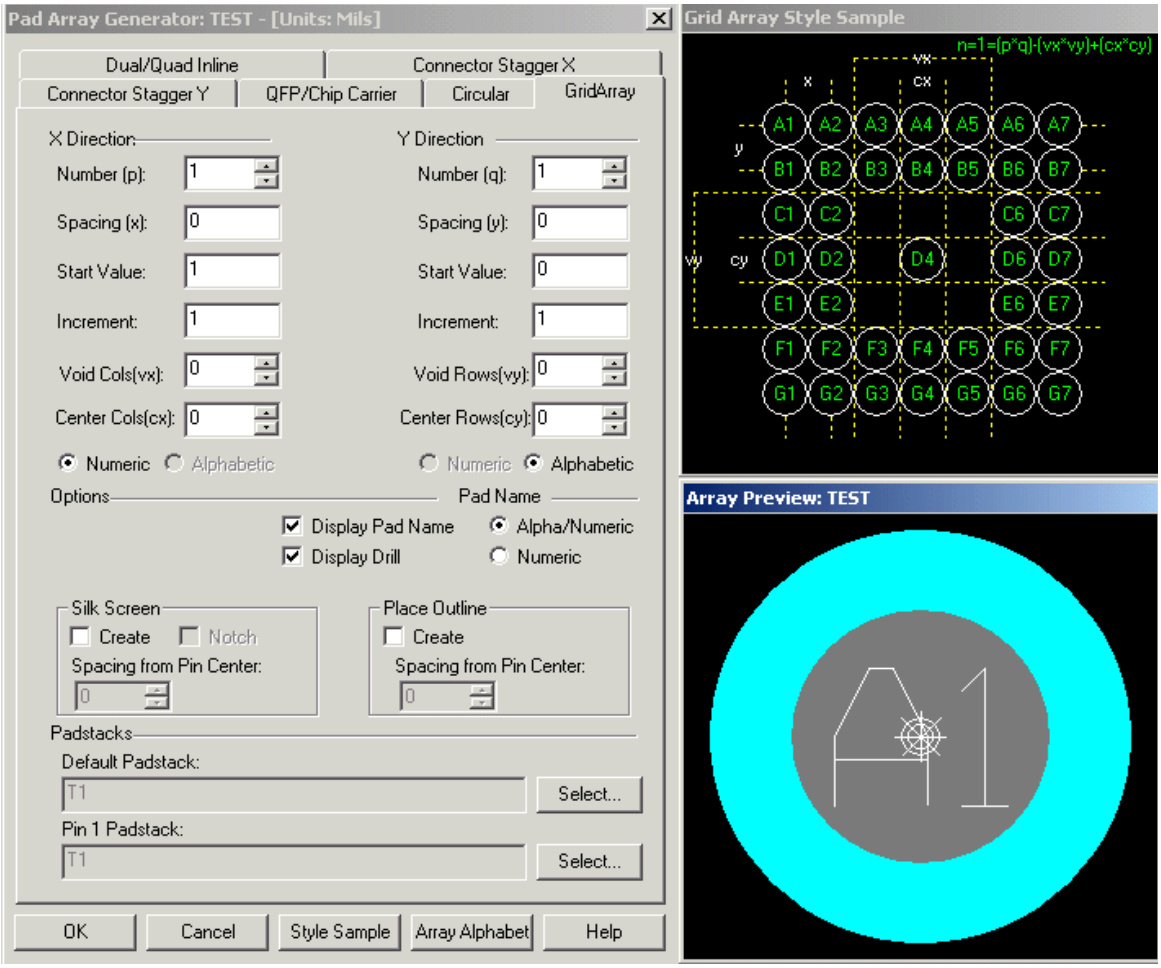


Figure 15-3 The Pad Array Generator dialog box, Preview Window, and Style Sample Window.

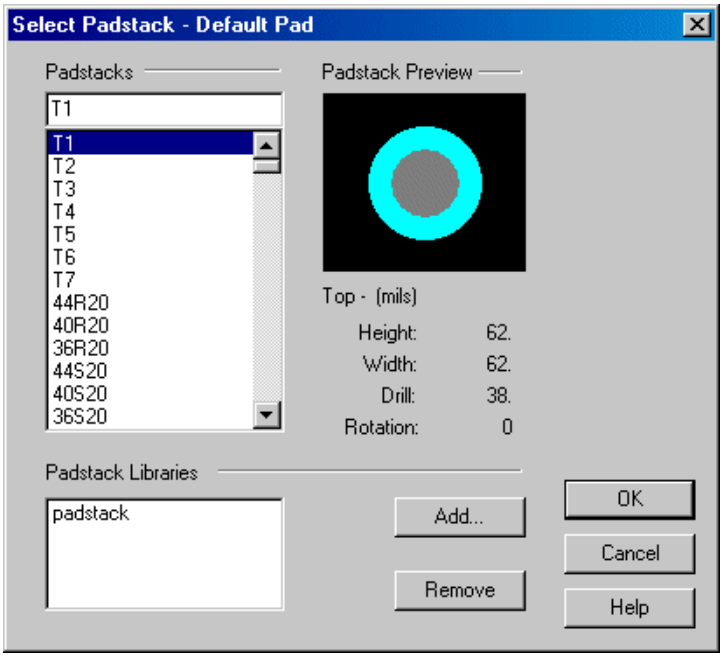
2 Choose the style of pad array to create by selecting the appropriate tab. There are six different styles from which to choose.

- **Dual/Quad Inline** - create an array that is limited to two columns in the X direction.
- **Connector Stagger X** - create an array that is numbered from left to right, top to bottom.

- **Connector Stagger Y** - create an array that is numbered from top to bottom, left to right.
- **QFP/Chip Carrier** - create a Quad Flat Pack or Chip Carrier array.
- **Circular** - create an array of pads in a circle.
- **GridArray** - create a Grid Array or Ball Grid array.

Note: To more efficiently choose an appropriate style, open the Style Sample window by pressing the Style Sample button. The Style Sample window displays spacing, stagger and other data that affect each pad array type. The sample is labeled with lower case letters that match the parameters in the style tabs.

3 In the Padstacks area, press the Select button to Select the default padstack. The Select Padstack dialog box opens.



4 Select an appropriate padstack in the Padstacks area and click OK.

5 If you need a different padstack for Pin 1 in this footprint, press Select to open the Select Padstack dialog box, and choose the appropriate padstack.

Note: Set the Pin 1 padstack only after you have set the default padstack.

6 Obtain spacing, stagger and other pad placement information from the part manufacturer's datasheet. Enter this data into X Direction, Y Direction and Options areas. For a description of each setting, see Pad Array Settings on the following pages.

Note: You must complete all modifications to the pad array settings before you click OK. If you prematurely generate the array, you may continue to edit pads individually in the footprint editor, otherwise you must start over by creating a new footprint.

7 Click OK to generate the pad array.

8 You can now continue creating your footprint in the footprint editor.

Pad Array Generator Settings

X Direction settings

Number (p)

Use this entry box to set the number of pad columns in the X direction.

Spacing (x)

Use this entry box to set the spacing between the pad column centers in the X direction. This value is entered in the selected units. For example, if you had a 60 mil pad and wanted a 20 mil space between the pads, you would enter 80. Total spacing cannot exceed 32 inches (812.8000 mm).

Start Value

Use this entry box to set the starting X value for the first pad in the array. The start value can be numeric or alphabetic; however, the start value must be entered in the entry box as a number. Zero corresponds to the letter “A” when using alphabetic pad labels.

Increment

Use this entry box to set the increment at which the pad label changes from the previous pad. For example, if you wanted to skip from 1 to 3 to 5 and so on, you would enter a value of 2.

Numeric

Select this option to use numeric pad labels for the X value.

Alphabetic

Select this option to use alphabetic pad labels for the X direction. You can specify what letters to use in the Array Alphabet dialog box.

Void Cols(vx)

Use this entry box to specify the number of columns that need to be removed from the center of a BGA pattern. This value should be in-synch with the value in the Void Rows entry box. You can see the effect in the Preview window.

This option is only available under the GridArray tab.

Center Cols(cx)

Use this entry box to specify the number of columns that need to be replaced in the center of a BGA pattern. This value should be in-synch with the value in the Center Rows entry box. You can see the effect in the Preview window.

This option is only available under the GridArray tab.

Y Direction settings

Number (q)

Number of pad rows in the Y direction.

Spacing (y)

Use this entry box to set the spacing between pad row centers in the Y direction. This value is entered in the selected units. For example, if you had a 60 mil pad and wanted a 20 mil space between the pads, you would enter 80. Total spacing cannot exceed 32 inches (812.8000 mm).

Start Value

Use this entry box to set the starting Y value for the first pad in the array. The start value can be numeric or alphabetic; however, the start value must be entered in the entry box as a number. Zero corresponds to the letter “A.”

Increment

Use this entry box to set the increment at which the pad label changes from the previous pad. For example, if you wanted to skip from 1 to 3 to 5 and so on, you would enter a value of 2.

Numeric

Select this option to use numeric pad labels for the Y value.

Alphabetic

Select this option to use alphabetic pad labels for the Y value. You can specify what letters to use in the Array Alphabet dialog box.

Void Rows(vr)

Use this entry box to specify the number of rows that need to be removed from the center of a BGA pattern. This value should be in-synch with the value in the Void Columns entry box. You can see the effect in the Preview window.

This option is only available under the GridArray tab.

Center Rows(cr)

Use this entry box to specify the number of rows that need to be replaced in the center of a BGA pattern.This value should be in-synch with the value in the Center Columns entry box. You can see the effect in the Preview window.

This option is only available under the GridArray tab.

Options settings

Stagger (w)

Use this entry box to set the amount every other pad in the left column is staggered from the column center. The allowed range is -10.0000 in. (-254.0000 mm) to 10.0000 in. (254.0000 mm).

Stagger (z)

Use this entry box to set the amount every other pad in the right column is staggered from the column center. The allowed range is -10.0000 in. (-254.0000 mm) to 10.0000 in. (254.0000 mm).

RowDelta (d)

Use this entry box to specify the addition or subtraction of a pad from a staggered row. A value of 1 adds a pad, and -1 removes a pad.

Radius (r)

Use this entry box to set the distance that the pads are from the center of the array. This value is entered in the selected units. The radius cannot exceed 32 inches (812.8000 mm).

Angle (q)

Use this entry box to set the angle between the current and next pad. This value is entered in degrees. The allowed range is -360˚ to 360˚. Negative values change the Style Sample window.

Note: If you have more pads than can be located in 360˚ with the current angle setting, the Pad Array Generator will continue to place pads around the circle and possibly overlap pads.

Display Pad Name

Select this option to display the pad names in the Array Preview window. To speed up display refresh rates, disable the Display Pad Name feature.

Display Drill

Select this option to display drill information in each pad in the Array Preview window.

Pin1 settings

Corner (QFP)

Select this option to set the location of Pin 1 at the corner of the pad array. Use this option when creating Quad Flat Packs. Changing the Pin 1 setting affects the Style Sample window.

Center (CC)

Select this option to set the location of Pin 1 at the top center of the array. Use this option when creating Chip Carriers.

Silk Screen settings

Use this group box to specify the silk screen spacing.

The Silk Screen group box on the Dual/Quad Inline tabbed page allows you to set the right spacing from the pin center. The one on the QFP/Chip Carrier tabbed page allows you to set the spacing from the center.

You can use the group box on each of the remaining tabbed pages to set the spacing from the pin center.

The Notch checkbox is also provided in the Silk Screen group box. If you select this check box, the spacing will always be set wrt pin 1.

Place Outline settings

Use this group box to draw the place outline. It allows you to set the spacing of the outline from the pin center.

Padstack settings

Default Padstack

Use the Select button to open the Select Padstacks dialog box, and select a padstack from the libraries. This sets the default padstack for all pads.

Pin 1 Padstack

Use the Select button to open the Select Padstacks dialog box, and select a padstack from the libraries. Use this setting to override the Default specified padstack for Pin 1.

Note: Set the Pin 1 padstack only after you have set the default padstack.

Adding pins to a footprint

Pins can be numeric, alphanumeric, and placed in any order. For example, you can name the pins 1, 7, 8, and 14 to fit a 4-pin oscillator that is numbered for a 14-pin part. Pin names *must* correspond to the pin numbers (or pin names if numbers are not used) of the schematic symbols.

Note: By default, Layout names the pins in numerical order beginning with the number 1. You must change the pin names in Layout to match the pin numbers in the schematic, or change them in the schematic library.

To add a pin to the footprint

1. In the Footprints list, select a footprint to which you want to add pins. The footprint appears in the footprint editor.
2. Click the pin toolbar button.
3. From the pop-up menu, choose New. A new pin attaches to the cursor.
4. Position the pin in the desired location and click the left mouse button to place the pin. As you move the pin, its X

and Y coordinates display in the status bar, so that you can use them as a guide for placing the pin.

Note: Numeric pin names are automatically renumbered when you add or delete pins in the Footprint Editor. If you do not want pin names to be automatically renumbered, deselect the Allow renumbering of pins check box in the User Preferences dialog box.

5. Press the INSERT key, then click the left mouse button to place each additional pin. The pins are placed using the distance established between pins 1 and 2.
6. To begin a new row of pins, select a pin and choose Copy from the pop-up menu.
7. Position the pin in the desired location and click the left mouse button to place the first pin of the new row.
8. Press the INSERT key, then click the left mouse button to place the second pin for the new row. This establishes the spacing for this row of pins.
9. Press the INSERT key, then click the left mouse button to place each additional pin. Continue placing pins until the footprint has the desired number of pins.

Assigning padstacks to footprint pins

Padstacks define the pins on each layer of the footprint. They possess properties on each layer of the board, such as shape and size. You can use the default padstacks included in the technology template, or define them when you are setting up the board. Once you define a padstack, you can assign it to pins in a footprint.

You can assign the same padstack to all the pins in the footprint using the Edit Footprint dialog box. Or, you can assign padstacks to individual pins using the Edit Pad dialog box. You can also input the exact coordinates for the pin location in the Edit Pad dialog box. This is a helpful tool for placing pins on a fine or irregular grid.

You can view the padstack assigned to each footprint pin in the Footprints spreadsheet (choose the spreadsheet toolbar button, then choose Footprints). You can view the padstack

definitions by layer for each padstack in the Padstacks spreadsheet (choose the spreadsheet toolbar button, then choose Padstacks).

To assign one padstack to all the pins of a footprint

1. Choose the spreadsheets toolbar button.
2. Select Footprints from the drop-down list.
3. In the Footprints spreadsheet, double-click on the footprint name.
4. Select a padstack from the Padstack Name drop-down list.
5. Choose the OK button to accept the setting and close the dialog box.

To assign a padstack to an individual pin

1. In the footprint editor, choose the Pin tool from the toolbar.
2. Press the SHIFT key and select the pin. From the pop-up menu, choose Properties.
3. Select a padstack for the pin from the Padstack Name drop-down list.
4. Choose the OK button to close the Edit Pad dialog box.

Attaching obstacles to footprints and pins

A variety of obstacles are used in the creation of footprints. In footprint libraries, the most commonly used obstacles are described below.

Place outlines

Layout's interactive and automatic placement utilities look for placement outlines. The outline is used to maintain a specified distance between parts. For surface mount parts, this outline should be large enough to provide sufficient space between

parts, thereby eliminating solder shadowing and facilitating the post-assembly inspection process.

Detail

Use detail obstacles to create silkscreen and assembly drawings for the parts. Assembly drawings represent the component shapes for manufacturing, and silkscreen references the actual parts on the board.

Copper

When copper is attached to a pin, it becomes an integral part of the pin. If the pin is moved, the copper moves with it. If the pin is attached to a net, then the copper automatically becomes a part of the net. When attached to a pin, copper can create a heat sink under a power part. Or, copper can create an odd-shaped pad for a special application.

Insertion outlines

An insertion outline is added to a footprint to represent the size of the auto-insertion head. It provides clearance around parts on the board so that the insertion machine head will not hit any components. This is also where you specify the height of the component.

Note: In the library manager, Layout assumes that the obstacles you create are to be attached to a pin of a footprint. For this reason, the Edit Obstacle dialog box supplies a Pin Attachment button instead of a Comp Attachment button when you're in the library manager. Although obstacles can be attached to pins, it is not a requirement.

To attach obstacles to footprint pins

1 In the library manager, create an obstacle as described in *Creating and editing obstacles*.

2 Select the obstacle and choose Properties from the pop-up menu. The Edit Obstacle dialog box appears.

3 Choose the Pin Attachment button. The Pin Attachment dialog box appears.

4 Select the Attach to pin option, supply the name of the pin in the Pin name text box, then choose the OK button.

5 Choose the OK button to close the Edit Obstacle dialog box.

Creating a place outline

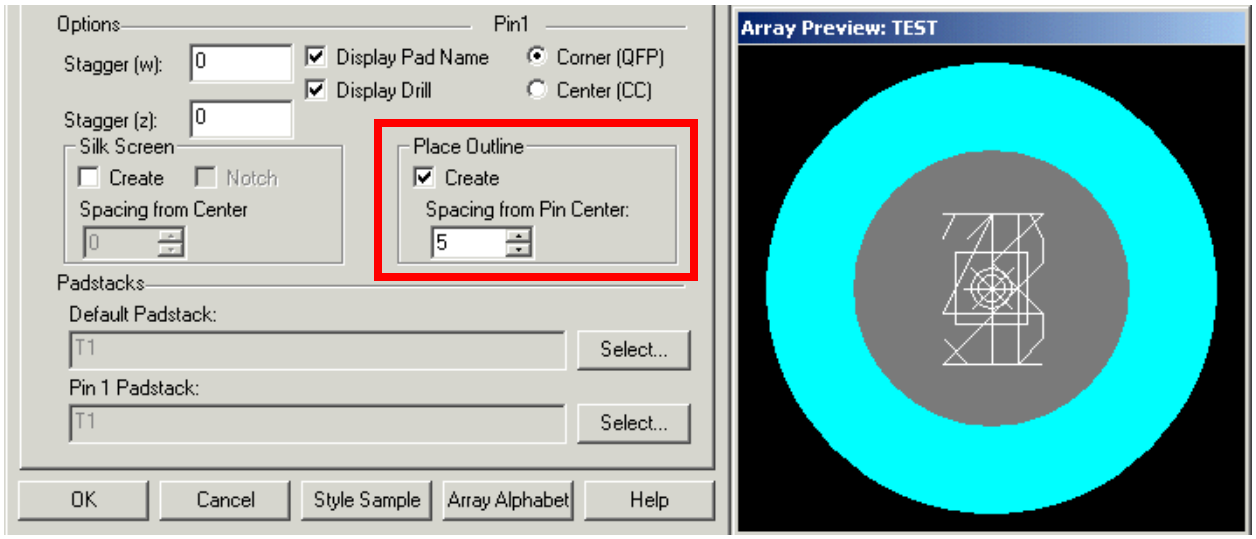
To create a place outline

A place outline defines the extent of the area that is reserved for a component's placement, and is used to maintain spacing between parts. Each footprint must have one. You can assign a height and specify the layer for a place outline. It can reside on the top layer for surface-mount parts, and on the global layer for through-hole parts.

Cadence recommends that you use only one place outline for a footprint. The height and shape of the place outline must closely represent the placement area required by a component. However, because Layout uses place outlines to calculate spacing violations, a place outline is usually rectangular so only a minimum number of calculations are necessary.

You can create a place outline either by using the Place Outline group box provided in the Pad Array Generator dialog box or by using the Obstacle toolbar button.

The Place Outline group box allows you to set the spacing of the outline from the pin center. Once you specify the spacing, the place outline is created automatically.



To create a place outline by using the Obstacle toolbar button:

1 Choose the Obstacle toolbar button.

Note: If you select the Show 3D Effects option in the User Preferences dialog box, and have assigned a height for a place outline, Layout displays a three-dimensional image representing the component's height, and indicates the height on the image.

2 From the pop-up menu, choose New.

3 Locate the point from which you want to start drawing the outline. There are three ways to move the cursor to this point: you can move the mouse, use the arrow keys, or press the TAB key to display the Find dialog box to go to the desired X, Y coordinates. Click the left mouse button once, or press the SPACEBAR. You will start drawing from this point.

Note: To place an obstacle at exact coordinates or coordinates that are off-grid, choose the Find toolbar button. In the Find coordinate or Component Name dialog box, enter the coordinates (X,Y) at which you want to place the first corner. Choose OK and repeat for the remaining corners.

If you are using a fine detail grid, use the mouse to approach the starting point, and then use the arrow keys to position the cursor. When you are at the starting location, click the left mouse button to start drawing the obstacle, or press the SPACEBAR to eliminate accidental mouse movement.

4. Double-click the left mouse button.

   The Edit Obstacle dialog box appears.

5. In the Obstacle Name text box, enter a name or leave the default number.

6. From the Obstacle Type drop-down list, select Place outline.

7. Optionally, in the Height text box, enter a value that corresponds to the height of the component.

8. From the Obstacle layer drop-down list, select the layer on which you want the place outline to appear.

9. Choose the OK button to accept the settings and close the Edit Obstacle dialog box.

10. Move from the starting coordinates to the desired location of the first corner. Click the left mouse button or press the SPACEBAR to insert the first corner. Move to the desired location of the next corner. Click the left mouse button or press the SPACEBAR to insert the second corner.

    Note: When you create an obstacle, such as a place outline, that is by definition an area, Layout automatically begins forming a closed area after you insert the first corner.

11. When you complete the final corner, press the F key, or from the pop-up menu, choose Finish. Layout automatically completes the place outline.

Note: A place outline cannot overlap another place outline.

Adding labels to footprints

You can assign several types of labels (reference designator, component value, user-defined custom properties, package name, and footprint name) to footprints in the footprint editor. You can specify which labels you want to assign using the Text Edit dialog box.

Note: For information on the Text Edit dialog box, and on creating labels, see Creating and editing text.

The labels in the footprint editor are placeholders preceded by ampersands (for example, *&Comp* or *&Value*) that are replaced by part properties from the schematic, such as reference designators and values.

Moving the insertion origin

Footprints have an *insertion origin* that serves as the location of the part, as specified in the insertion report.

To move the insertion origin

1. In the library manager, from the Tool menu, choose Dimension, then Move Datum.
2. Move the cursor to the target location for the insertion origin. (Be careful not to click the mouse button, because the datum will move to the location.)
3. From the pop-up menu, choose Move Insertion Origin.
4. Click the left mouse button on the screen to place the insertion origin at that location.
5. From the pop-up menu, choose End Command.

To center the insertion origin

1. In the library manager, from the Tool menu, choose Dimension, then Move Datum.
2. From the pop-up menu, choose Center Insertion Origin. The insertion origin centers itself within the footprint.

3 From the pop-up menu, choose End Command.

Centering footprint insertion origins

Use this command to specify the insertion origin of a footprint in order to direct the efforts of a pick-and-place machine for board manufacture. Use this command to move the datum, or insertion origin, to the center of the footprint. The footprint center is calculated with respect to the pads.

To center the insertion origin of a footprint

1 In the library manager window, select the footprint.

2 From the Tool menu, point to Dimension and choose Move Datum.

3 From the pop-up menu, choose Center Insertion Origin. Layout places the insertion origin at the center of the footprint.

Related topics

Move Datum command

Edit Footprint dialog box

Editing footprints and footprint pins

You can edit footprints in the footprint editor. Or, you can edit footprint data using the Footprints spreadsheet. One method may be more practical than the other, depending on the type of activity you are performing. Typically, when editing obstacles or text, you use the footprint editor. When editing multiple pin locations or padstacks, you use the spreadsheet.

You can edit all the pins of a footprint at once, or edit individual pins. You can modify the location, padstack assignment, and entry and exit rules of a pin. Additionally, you can make a pin a forced or preferred thermal relief, and allow vias to be placed under the pin.

The Edit Footprint dialog box and the Edit Pad dialog box offer the same editing options. However, changes you make in the Edit Footprint dialog box affect all of the pins in the footprint, whereas changes you make in the Edit Pad dialog box affect only the selected pin.

To edit footprint pins in the footprint editor

1. In the library manager, choose the pin toolbar button.
2. Double-click on a pin. The Edit Pad dialog box appears.
3. Edit the settings as desired, then click OK.

To copy, move, delete, or rotate pins in the footprint editor:

1. Click on the pin you want to copy, move, delete or rotate.

   To select a group of pins in the Footprint Editor, do one of the following:

   - Press the left mouse button and drag it over the pins you want to include in the selection
   - Press the CTRL key and click on the pins you want to include in the selection.

   To deselect a pin in the selection, press the CTRL key and click on the pin.

- Press CTRL+C to copy the selected pins.
- Drag the mouse to move the selected pins.

  Position the pin in the desired location and click the left mouse button to place the pin. As you move the pin, its X and Y coordinates display in the status bar, so that you can use them as a guide for placing the pin.

- Press the Delete key to delete the selected pins.

  Note: Numeric pin names are automatically renumbered when you add or delete pins in the Footprint Editor. If you do not want pin names to be automatically renumbered, deselect the Allow renumbering of pins check box in the User Preferences dialog box.

- From the Tool menu point to Pin and choose Rotate to rotate the selected pins.

To edit the footprint or footprint pins using the spreadsheet

1. Click the spreadsheet toolbar button, then choose Footprints.
2. To edit all the pins in the footprint, double-click on the footprint name. The Edit Footprint dialog box appears.
   or
   To edit a footprint pin, double-click on a pad name. The Edit Pad dialog box appears.
3. Edit the settings as desired, then click OK.

Editing and copying padstacks

Editing padstacks

You can edit the default padstack definitions predefined in Layout, or padstacks that you have defined while setting up the board. Editing changes you make in the Edit Padstack dialog box are applied to all layers of the padstack. Editing changes you make in the Edit Padstack Layer dialog box are only applied to the selected layer.

To edit a padstack on all layers

1. Choose the spreadsheet toolbar button, then choose Padstacks. The Padstacks spreadsheet appears.
2. Double-click on the padstack name. The Edit Padstack dialog box appears.
3. Edit the settings as desired (choose the dialog box's Help button for information on the options in the dialog box), then choose the OK button.

To edit a padstack on selected layers

1. Choose the spreadsheet toolbar button, then choose Padstacks. The Padstacks spreadsheet appears.
2. Double-click on a layer name. The Edit Padstack Layer dialog box appears.
3. Edit the settings as desired (choose the dialog box's Help button for information on the options in the dialog box), then choose the OK button.

Copying padstack layers

You can add a new layer to your board and copy the padstacks from an existing layer to the new layer.

To copy a padstack layer

1. In the design window, choose the spreadsheet toolbar button, then choose Layers. The Layers spreadsheet appears.
2. Select an unused layer (for example, INNER3) and designate it as a used layer (for example, as a plane layer).
3. Choose the spreadsheet toolbar button, then choose Padstacks. The Padstacks spreadsheet appears.
4. Click in the left most title cell to select all items, then choose Copy Layer from the pop-up menu. The Copy Padstack Layer dialog box appears.
5. Select appropriate layers from the Source Layer and Target Layer drop-down lists (for example, GND as the source layer and INNER3 as the target layer), then choose the OK button. The new layer inherits the padstacks.

Copying padstacks into a padstack library

You can copy a padstack from a board or library file to a custom padstack library, copy a padstack from a footprint to a

custom padstack library, or copy a padstack from one padstack library to another.

Note: OrCAD provides a padstack library named PADSTACK.LLB with Layout. Because any custom padstacks added to PADSTACK.LLB are overwritten when you receive a new version of Layout software, you should save any custom padstacks you create in a custom padstack library.

To copy a board or library file padstack to a custom padstack library

1. Open a board or library file in the design window.
2. Choose the Spreadsheets toolbar button, then choose Padstacks. The Padstacks spreadsheet appears.
3. Select the padstack you want to copy.
4. From the Tool menu, point to Padstack and select Save to Library. The Save padstacks - Select library dialog box appears.
5. Select a library from the Select Library drop-down list, or click the Browse button to browse for a library name,

   or

   create a new library by clicking the Create New button and defining a path and file name for the new library.

To copy a padstack from a footprint to a custom padstack library

1. From the File menu, choose Library Manager, or click the Library Manager toolbar button. The library manager window displays.
2. In the Libraries group box, select or add an appropriate library.
3. In the Footprints group box, select an appropriate footprint. The footprint displays.
4. Choose the Pin toolbar button.

5 Hold down the SHIFT key and select the padstack you want to copy to your custom padstack library.

6 Choose the Spreadsheets toolbar button, then choose Padstacks. The Padstacks spreadsheet displays, with the padstack you selected highlighted.

7 From the Tool menu, point to Padstack and select Save to Library. The Save padstacks - Select library dialog box displays.

8 Select a library from the Select Library drop-down list, or click the Browse button to browse for a library name,

or

create a new library by clicking the Create New button and defining a path and file name for the new library.

To copy a padstack from one custom padstack library to another

1 Open the first custom padstack library in the design window.

2 Choose the Spreadsheets toolbar button, then choose Padstacks. The Padstacks spreadsheet displays.

3 Select the padstack you want to copy.

4 From the Tool menu, point to Padstack and select Save to Library. The Save padstacks - Select library dialog box displays.

5 Select a library from the Select Library drop-down list, or click the Browse button to browse for a library name,

or

create a new library by clicking the Create New button and defining a path and file name for the new library.

The Catalog Tool

Catalog files are .MAX files that contain all of the components in a library. The components are sorted and displayed on

multiple catalog pages in each .MAX file. These files can be easily printed with the Print Catalog tool. If you own Adobe Acrobat you can also create .PDF files of your catalog.

Creating a catalog

Catalog files are MAX files that contain all of the components in a library. Components are sorted and displayed on pages in each catalog file. Since catalog files are merely MAX files, they can be easily edited, annotated, organized and printed. There can be multiple catalog pages within each MAX file.

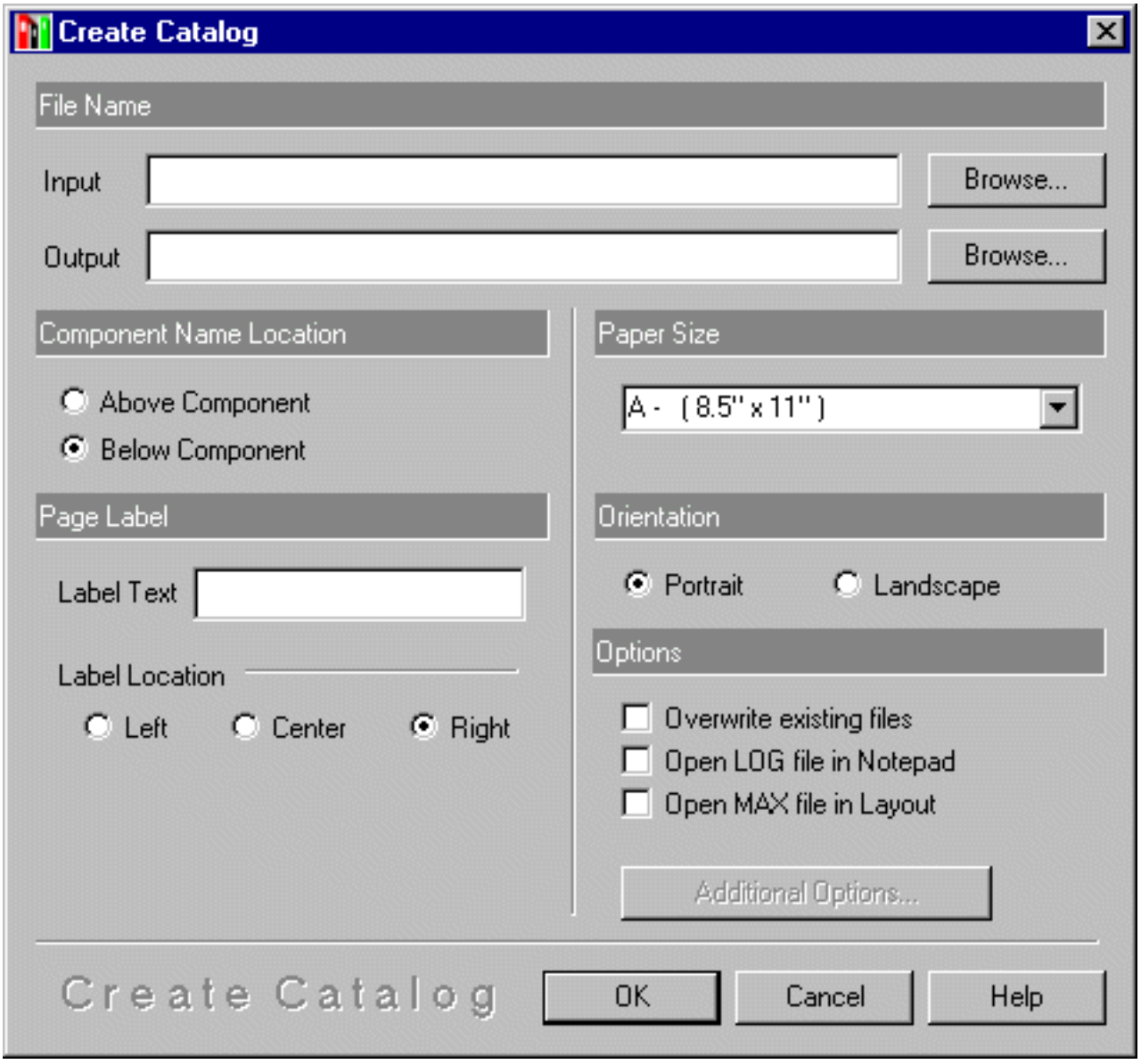
Catalog files can be created from either library (LLB) or board (MAX) files.

Note: When creating a catalog file from a MAX file, before you create the catalog it is recommended that you run Cleanup Design with the Remove Unused Footprints option selected. This removes any lingering footprints from the database that are not currently used in the design.

To create a catalog file

1 Open the Layout session window

From the Tools menu, point to Catalog and choose Create. The Create Catalog dialog box appears.



2 In the Input text box, enter the filename and path to the file to catalog, or click the Browse button to locate the file. You can use a library (LLB) or board (MAX) file to create a catalog.

3 In the Output text box, enter a name for the catalog, or accept the default name.

4 In the Component Name Location area, choose where you want the component name to display.

5 Enter text that you want at the top of each page into the Page Label text box. Note that each page label automatically includes a page number.

6 Choose a location option for the page label.

7 Choose a paper size and orientation.

8 Click the Additional Options button. Enter any spacing, sorting or margin setting in the Additional Options dialog box and click OK. For detailed information about each option, see the Create Catalog dialog box help topic.

9 Click OK to create the catalog file.

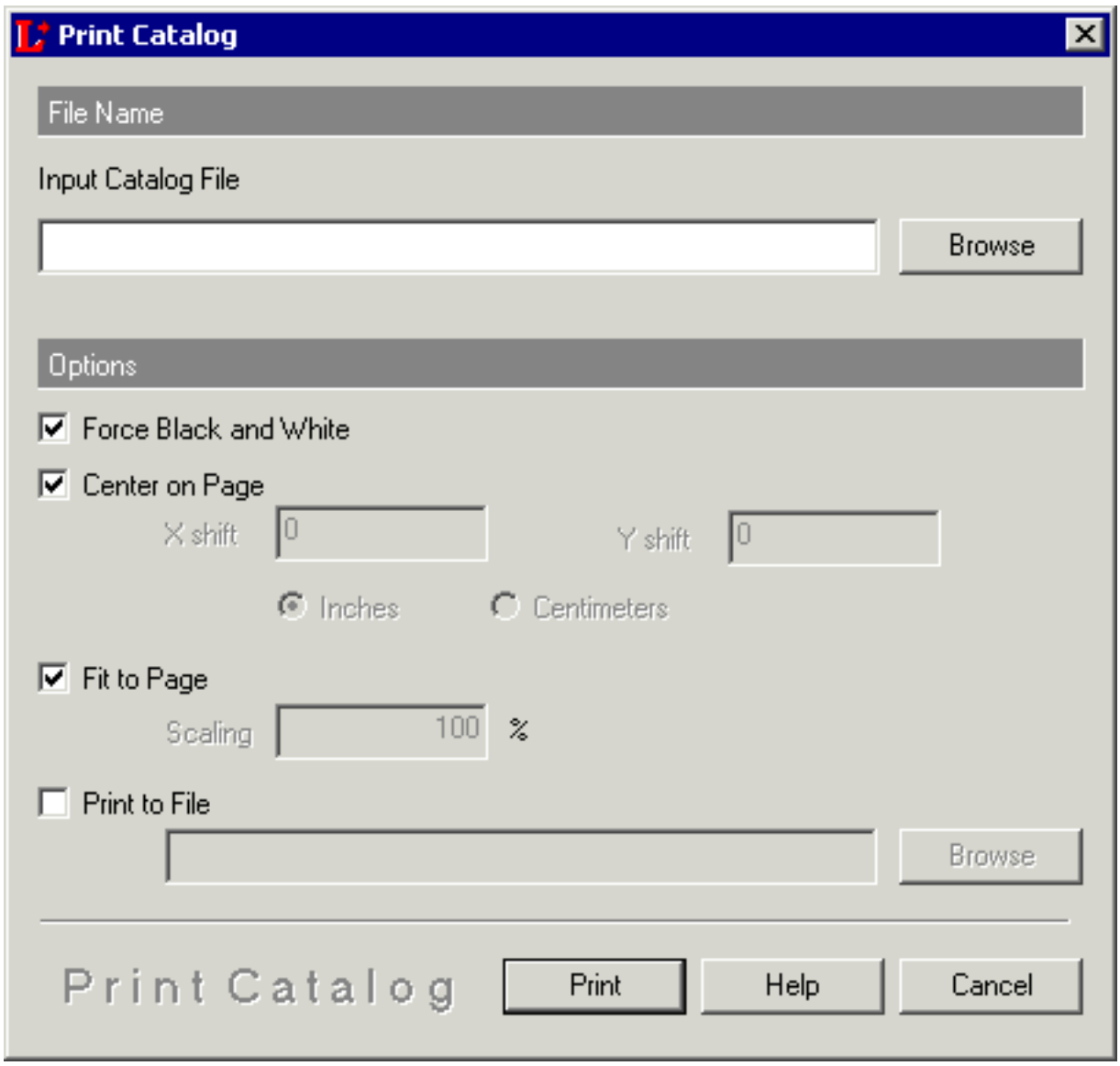
Printing a catalog

To properly print a catalog file, you must use the Print Catalog tool and not the typical Print command in Layout.

To print a catalog file

1 Open the Layout session window.

2 From the Tools menu, point to Catalog and choose Print. The Print Catalog dialog box appears.



3 In the Input Catalog File text box, enter the filename and path to the catalog file, or click the Browse button to locate the file. Only enter a MAX file that was created as a catalog file. Other board files cannot be printed with this tool.

4 In Options area, choose any print options that you require. For detailed information about each option, see the Print Catalog dialog box help topic.

5 Click Print. The Windows Print Manager appears.

6 Set any options necessary for your printer and click Print.

If you have a very large printer or plotter, you can use the normal Print command in Layout to print an entire catalog MAX file on one sheet. The Print command in Layout prints the catalog file as if it were a typical MAX file.

Related topics

Creating an Acrobat PDF file of an OrCAD Layout Library

After you have created a catalog file, you will want to share that file with others. An Adobe Acrobat PDF file is a good format to use for sharing your catalog electronically.

To create a PDF file, you must own a copy of Adobe Acrobat. Please note that this is not the free Acrobat Reader, but the full version of Adobe Acrobat. For more information about purchasing Acrobat, please contact Adobe at www.adobe.com.

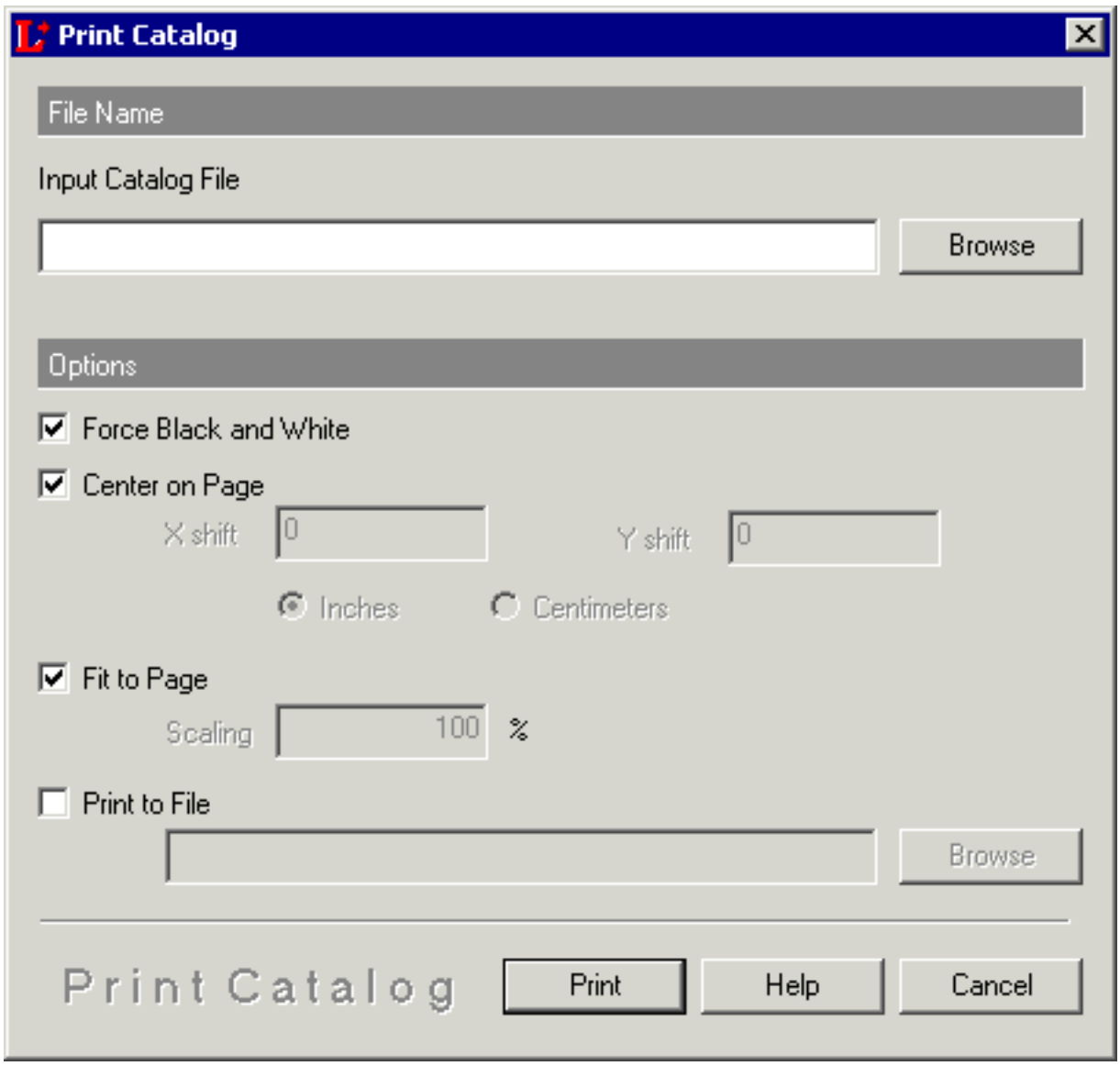
Note: The following procedure was developed with Adobe Acrobat 5.0 on Windows 2000. If you are using a different version of Acrobat or Windows, options and dialog boxes may vary slightly.

To create an Acrobat PDF of an OrCAD Layout Library

1 Create a catalog file of the library. For detailed directions, see the help topic Creating a Catalog.

2 Open the Layout session window.

3 From the Tools menu, point to Catalog and choose Print. The Print Catalog dialog box appears.

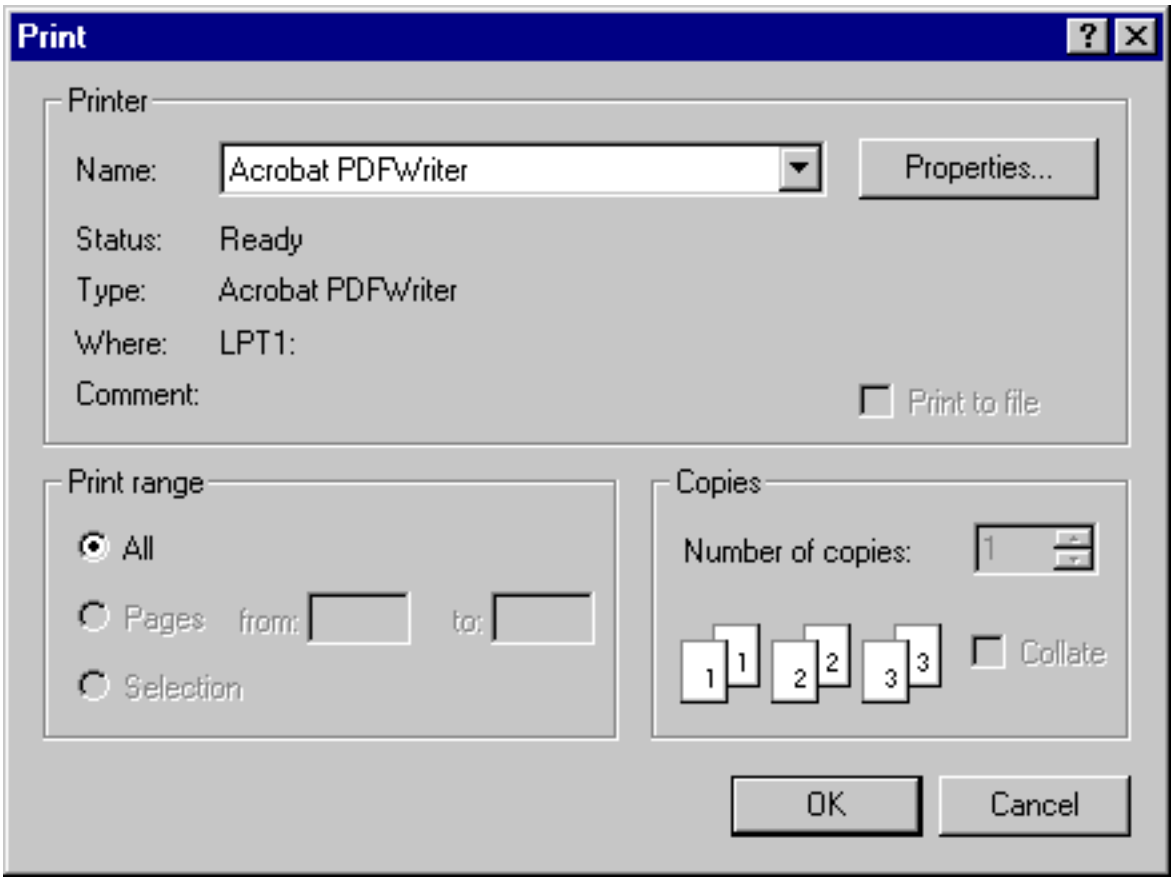


4 In the Input Catalog File text box, enter the filename and path of the catalog to print. Only enter a MAX file that was created as a catalog file. Other board files cannot be printed with this tool.
or
Click the Browse button to locate the file.

5 In the Options area, choose the Force Black and White option. On the white background of PDF files, colors can be difficult to view. If you would like to use color in your PDF, it is recommended that you change the colors in the Colors Spreadsheet to more visible darker colors.

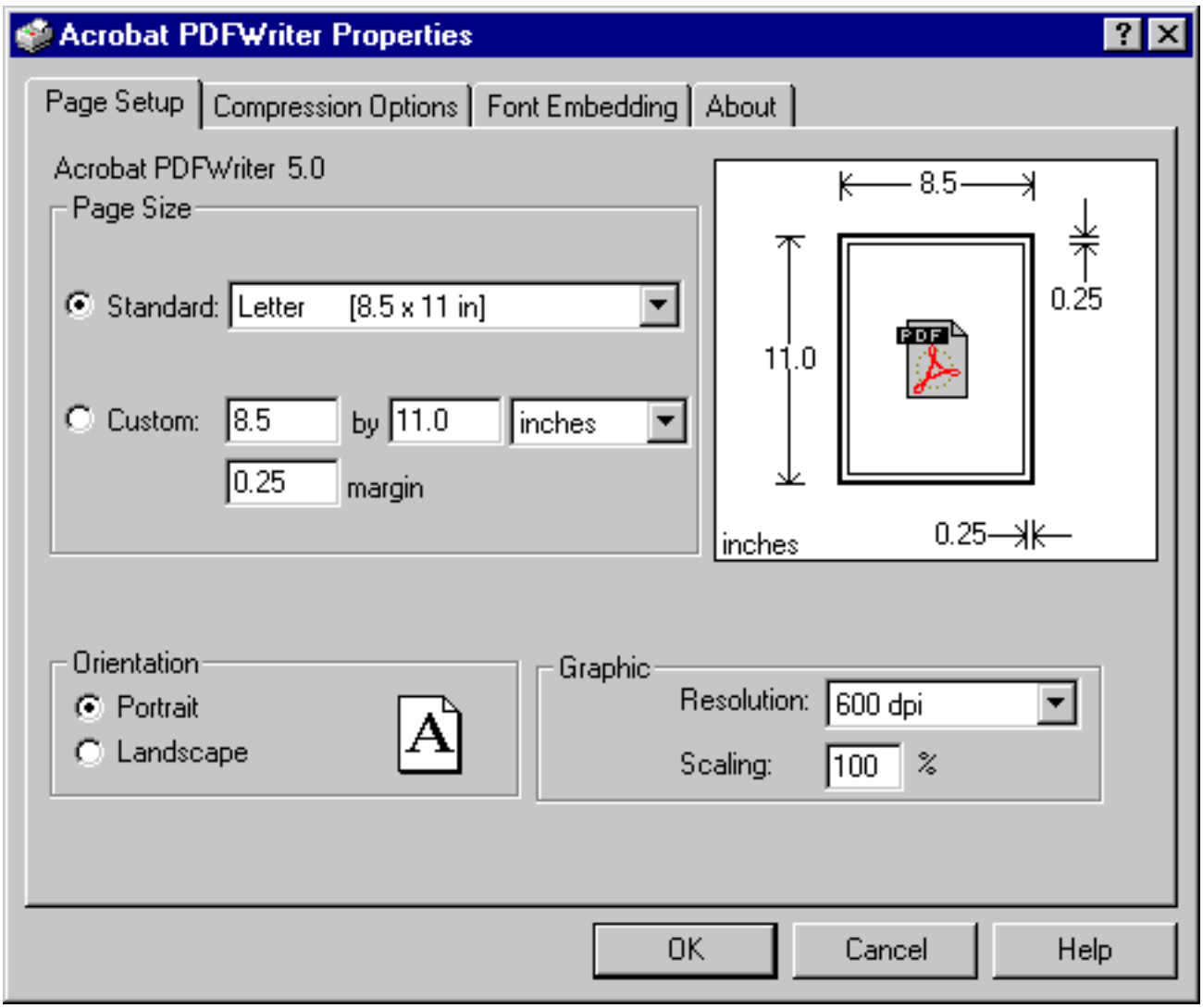
6 Choose any other desired options. For detailed information about each option, see the Print Catalog dialog box help topic.

7 Click Print. The Windows Print manager appears.



8 Choose Acrobat PDFWriter from the drop down menu.

9 Click the Properties button to open the Acrobat PDFWriter Properties dialog box.



10 For the highest quality output for print as well as on-screen viewing, choose the following settings:

- On the Page Setup tab, in the Graphic group box, choose a resolution of 600 dpi. This dramatically improves print quality, and does not affect file size significantly.
- On the Compression Options tab, click the Default button. This resets all compression options, and gives you a good final product.

11 Click OK to accept the new PDFWriter settings.

12 Click OK in the Windows Print Manager.

Creating design libraries

Design libraries are libraries that contain all the footprints required for a design.

If you use a design library, you can limit the AutoECO process to only the design library. This saves time in the design cycle as the AutoECO process needs to scan only the design library instead of scanning all the libraries made available for use in Layout. For more information, see Using design libraries in AutoECO on page 40. For more information on making a library available for use in Layout, see Making libraries available for use on page 531.

Another need for design libraries arises because the footprint libraries provided with Layout or your custom footprint libraries may be updated from time to time. This may cause situations where your design references footprints that have changed and, thereby, impact the functionality of the design. To avoid this, you can use design libraries.

Using design libraries also helps in archiving the design. As all the footprints used in the design exist in a single library that is located in the project directory, you can easily archive your design. If you do not use a design library, archiving your design will require you to include all the footprint libraries that contain the footprints referenced by your design.

To create a design library

1 In the Layout session window, from the Tools menu, choose Configure Design Library.

The Configure Design Library dialog box appears.

2 Enter the path and filename of the netlist (.MNL) file for the design or click the Browse button to select the file.

3 Enter the path and filename for the design library you want to create.

4 Select the unit (English or Metric) in which you want the footprints to be stored in the design library.

5 Specify the footprint search criteria by doing the following:

- ❑ Select the Footprint name check box if you want find only the footprints that have the same name as the footprint specified for the part in your OrCAD Capture schematic (using the PCB Footprint column in the property editor in Capture), or where the footprint is not specified, the part name in your Capture schematic.
- ❑ Select the Number of Pins check box if you want to find only the footprints that have an equal or greater number of pins as the part in your Capture schematic and also have the same pin names as the pin names on the part in your Capture schematic.

For information on how Layout selects the footprints to be added to the design library, see How Layout selects footprints to be added to a design library on page 574.

6 Select the Add configured libraries check box if you want all the libraries you have made available for use in Layout to be searched for footprints you want to add to the design library.

The libraries you have made available for use in Layout are displayed. For more information on making libraries available for use in Layout, see Making libraries available for use on page 531.

7 Do the following to add or remove the libraries you want to search for footprints:

Click	To
Add to List	Add other libraries to the list. **a.** Select a library. **b.** Click the Open button to add the library to the list of libraries. **Note:** You can add a maximum of 250 libraries in the list.
Remove	Remove the selected library from the list.
Remove All	To remove all the libraries displayed in the list.

Note: Adding or removing libraries in the list does not mean that the libraries are added to, or removed from the list of libraries made available for use in Layout. You add or remove libraries in the list only for specifying the libraries you want to search for footprints. For more information on making libraries available for use in Layout, see Making libraries available for use on page 531.

8 Click the Search button.

The footprints specified for the parts in your OrCAD Capture schematic (using the PCB Footprint column in the property editor in Capture), or where the footprint is not specified, the part name in your Capture schematic are displayed in the Symbol/Capture Footprint list. The footprints selected for adding to the design library are displayed next to each Capture footprint or part name in the Footprint list.

- ❑ If no footprint is found for a part, the text “-- Not Found --” in red color is displayed next to the part in the Footprint column.

If a footprint is not found for a part, double-click on the row for the part. The Search Footprint dialog box appears. Search for the footprint and click Add to select the footprint for the part.

- If more than one footprint is found for a part, the Select Footprint dialog box appears. The parts for which multiple footprints were found are displayed in the Symbol/Capture Footprint list.

  Click the drop-down list next to each part to select the required footprint, then click OK to close the Select Footprint dialog box.

  Note: If you later want to change the footprint for a part for which multiple footprints were found, double-click on the row for the part to display the Replace Footprint (footprint name) dialog box. Select the required footprint and click Replace.

- If you want to change the footprint selected for a part, select the row for the part and click the Replace Selected Footprint button. The Replace Footprint dialog box appears. Perform a search for the footprint, select the footprint in the Footprint list and click Replace.

For information on how Layout selects the footprints to be added to the design library, see How Layout selects footprints to be added to a design library on page 574.

9 Click the Save button to save the footprints to the design library.

Note: Before saving the footprints to the design library, ensure that every Capture part is mapped to a corresponding footprint. If any Capture part is not mapped to a footprint, errors will be displayed when you run AutoECO with the design library selected (select the Use design library only check box in the AutoECO dialog box and specify a design library).

Note: If you save the selected footprints to an existing design library, Layout prompts you if:

- a footprint with the same name already exists in the design library. You can replace the existing footprint with the new footprint.
- a footprint is already mapped to a Capture part. You can update the mapping information.

Tip

OrCAD recommends that you create one design library for each design.

How Layout selects footprints to be added to a design library

When you create or update a design library, Layout scans the netlist (.MNL) file for the design to select the footprints required for the design.

- If you select footprint name as the search criteria (select the Footprint Name check box in the Replace Footprint (footprint name) dialog box, the following procedure is used to select the footprints for the design:

  a. If a footprint name is specified for a part in your OrCAD Capture schematic (using the PCB Footprint column in the property editor in Capture), that footprint is selected.

  b. If a footprint name is not specified for a part in your OrCAD Capture schematic, the footprint that is linked to the part name in the USER.PRT file and the SYSTEM.PRT file is selected. For more information on the USER.PRT and SYSTEM.PRT files, see USER.PRT on page 123 and SYSTEM.PRT on page 122.

  Note: Layout looks first in the USER.PRT file, then in the SYSTEM.PRT file to identify the footprint. If a part name is linked to one footprint in the USER.PRT file and to some other footprint in the SYSTEM.PRT file, Layout uses the information in the USER.PRT file.

c. If no footprint name is specified for a part in OrCAD Capture and no footprint is linked to the part name in the USER.PRT or SYSTEM.PRT file, the footprint whose name is the same as the name of the part in your OrCAD Capture schematic is selected.

- If you select number of pins as the search criteria (select the Number of Pins check box in the Configure Design Library dialog box), Layout selects footprints that have an equal or greater number of pins as the part in your Capture schematic and also have the same pin names as the pin names on the part in your Capture schematic.

Visual CADD

16

Visual CADD is used in the Layout design flow to translate mechanical drawings.

When you translate a printed circuit board from another PCB design program for use in OrCAD Layout, it is always best to use a translator specifically created for this purpose. If such a translator is not available, or if the board was actually designed in AutoCAD, it is possible to reconstruct the board from one or more DXF files.

Visual CADD is only supported by OrCAD for interactive DXF translation. To access Visual CADD, click the Interact button on the DXF to MAX dialog box.

For all other mechanical drawing needs, please use IntelliCAD. To launch IntelliCAD, from the session window, choose IntelliCAD from the Tools menu.

Manipulating the MECHANICAL mappings

If your mechanical engineering group placed the board outline on a DXF layer called BOARD_OUTLINE, then the translation will proceed normally. If they placed it on some other layer (PERIMETER, in this example), change the appropriate line in MAXDXF.INI to read:

```
BOARD_OUTLINE = PERIMETER
```

If there is a keepout zone on DXF layer 2, then change the appropriate line in MAXDXF.INI to read:

```
COMP_KEEPOUT_ALL = 2
```

Any plated mounting holes stored in the DXF as circles drawn on layer HOLES would require a line in MAXDXF.INI such as:

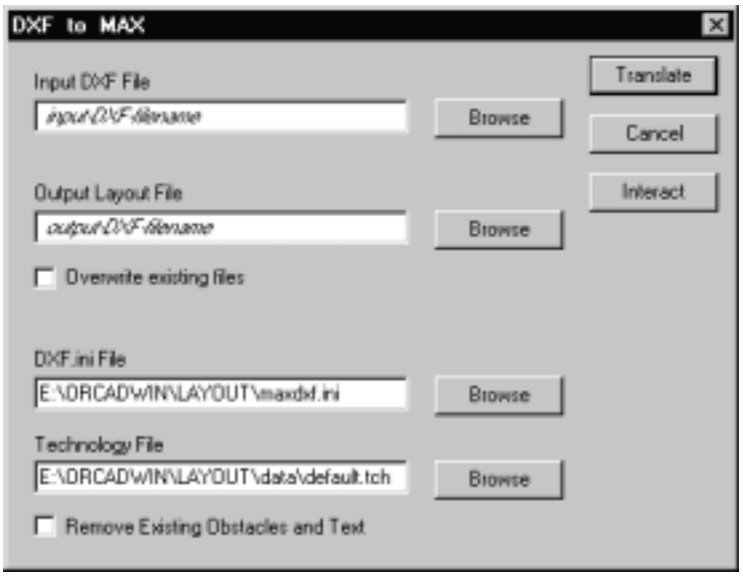
```
PLATED_MOUNTING_HOLES = HOLES
```

Interactive DXF translation

If all the objects are on one DXF layer, translation is more complicated than Automatic translation. MAXDXF cannot distinguish one kind of object from another, but you can use interactive translation to filter groups of objects for MAXDXF.

To translate objects interactively

1. Open the Layout session frame.
2. From the File menu, choose Import, then choose DXF to MAX.



3. In the upper pair of text boxes, type the names of your input DXF file and the output Layout file (or accept the default output filename). You can accept the default filenames in the lower pair of text boxes.

4 Choose the Interact button. The screen clears, then Visual CADD opens displaying the input DXF file. It may take a few moments for Visual CADD to process all the information in the DXF file.

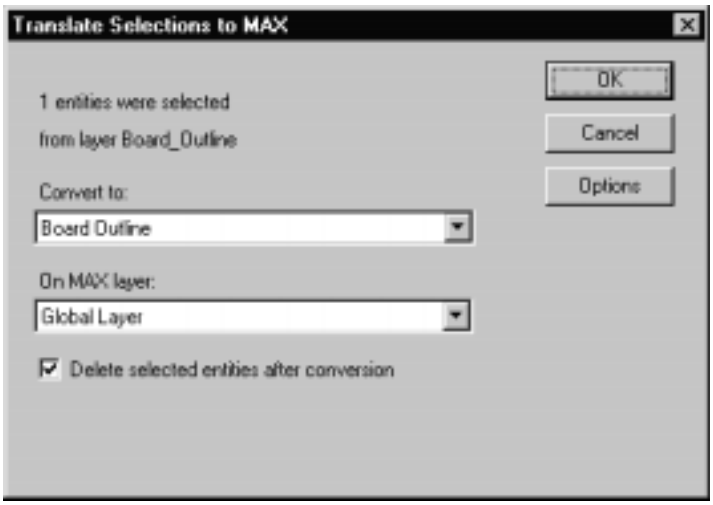
5 Choose the Selection tool from the tool palette at the left edge of the window, then select the board outline. If the board outline is built up of individual lines and arcs, follow these steps to select the entire outline:

   a. From the pop-up menu, choose Select By, then choose Adjoining.

   b. Select any part of the board outline.

   c. Inspect the selected outline. Add lines and arcs to the selection set, or remove them, as needed.

6 From the tool palette at the left edge of the window, choose the Convert to MAX tool.



7 From the drop-down lists, select Board Outline and Global Layer.

8 Select the Delete selected entities after conversion option. This will help you keep track of your progress as you translate the DXF.

9 Choose the OK button.

10 Repeat this process for other objects you want to translate.

11 When you finish, exit Visual CADD but do not save the modified DXF file. If you have been deleting objects as your translate them, the DXF file is mostly empty at this point, and saving it would replace your existing file with the empty version. The translation process completes, and Notepad displays any warning or error messages generated.

Importing the DXF file

In Layout, all tracks must begin and end on a pad, or they must end with a pending unrouted connection to a pad. This means that you must translate component pads *before* tracks. Vias, which must be circle objects (not simply four arcs), are always translated as "through" vias.

Simple circular component pads in a DXF file are usually defined as circle objects, while oval or rectangular pads are composed of several lines, arcs, or polylines grouped as a Visual CADD symbol (equivalent to an AutoCAD *block*).

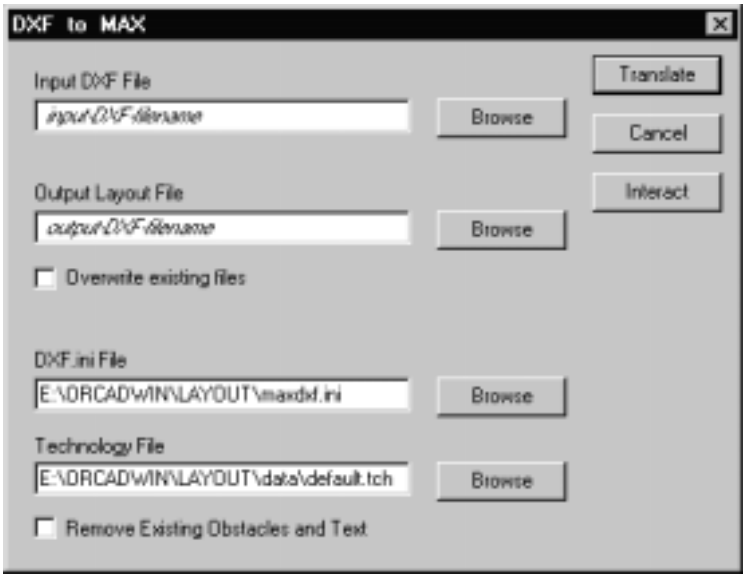
Note: If you have only DWG files, you can load them into IntelliCAD and save them as DXF for interactive translation.

The following procedures use the term *component pads*, regardless of how they are represented in your DXF file.

To translate the board outline and any zones

1 Open the Layout session frame.

2 From the File menu, choose Import, then choose DXF to MAX.



3 In the upper pair of text boxes, type the names of your input DXF file and the output Layout file (or accept the default output filename). You can accept the default filenames in the lower pair of text boxes.

4 Choose the Interact button. The screen clears, then Visual CADD opens displaying the input DXF file.

5 Translate the board outline as described in the Interactive translation section.

Note: Remember to leave the Delete selected entities after conversion option selected in the Translate Selections to MAX dialog box.

6 One at a time, translate any zones (keepout, keepin, copper pour, and so on) by the same process.

Translating components

One at a time, translate each component by setting the origin, translating the component pads, and then translating the silkscreen lines and text.

To set the component origin

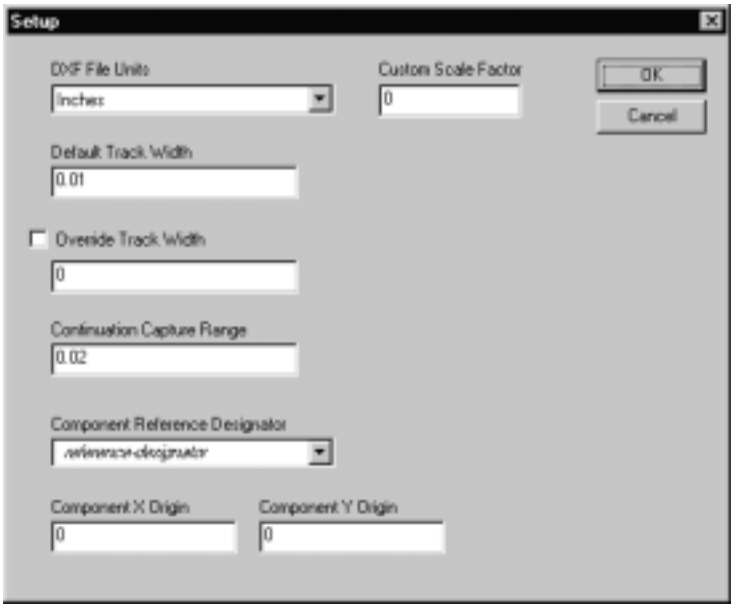
1 If your DXF is hierarchical and your component is a symbol unto itself, simply select the component symbol.

or

If your DXF *is not* hierarchical or your component *is not* a symbol unto itself, then from the pop-up menu choose Select By, then choose Window, and draw a box around the component pads.

2 From the tool palette at the left edge of the window, choose the Convert to MAX tool.

3 From the upper drop-down list, select Set Component Origin, then choose the Options button.



4 In the Component Reference Designator text box, type a unique name. Check the names in the drop-down list to make sure the name you enter is unique.

5 Choose the OK button in both of the open dialog boxes.

Note: Even though the Delete selected entities after conversion option is still selected, the translator does not delete any object used only to set the component origin.

To translate component pads

1. If your DXF is hierarchical and your component is a symbol unto itself, select the component symbol, then type EX to explode it.
2. From the pop-up menu choose Select By, then choose Window, and draw a box around the component pads. The conversion is limited to circles and symbols, so don't worry if you accidentally select portions of tracks or silkscreen-they will be ignored.
3. From the tool palette at the left edge of the window, choose the Convert to MAX tool.
4. From the upper drop-down list, select the appropriate type of conversion:
   - ❑ Circles to SMT Pads
   - ❑ Symbols to SMT Pads
   - ❑ Circles to THRU Pads
   - ❑ Symbols to THRU Pads
5. If you are translating SMT Pads, then select either TOP or BOTTOM from the On MAX layer drop-down list.
6. Make sure the Delete selected entities after conversion option is selected, then choose the OK button.

To translate the component silkscreen lines and text

1. From the pop-up menu, choose Select By, then choose Window, and draw a box around the component silkscreen. If the selection set includes tracks, just use the Layer Manager to hide the track layers.
2. From the tool palette at the left edge of the window, choose the Convert to MAX tool.
3. From the upper drop-down list, select Component Silkscreen.
4. From the lower drop-down list, select either SSTOP or SSBOT.

5 Make sure the Delete selected entities after conversion option is selected, then choose the OK button.

Translating remaining elements

Next, translate board silkscreen elements and tracks.

To translate silkscreen text

1 From the pop-up menu, choose Filter.

2 Set Entity to Text, and Layer to the silkscreen top layer for this component.

3 Make sure Color and Font are set to *All*.

4 Select the Filter option, and choose the OK button.

5 From the pop-up menu, choose Select By, then choose All.

6 From the tool palette at the left edge of the window, choose the Convert to MAX tool.

7 From the upper drop-down list, select Text.

8 From the lower drop-down list, select SSTOP.

9 Make sure the Delete selected entities after conversion option is selected, then choose the OK button.

10 Repeat steps 1 through 9 for the silkscreen bottom layer, selecting SSBOT at step 8.

To translate silkscreen lines

1 From the pop-up menu, choose Filter.

2 Deselect the Filter option, and choose the OK button.

3 Use the same process to translate your board silkscreen lines, but convert them to Free Track (step 7) on the SSTOP or SSBOT layer (step 8).

To translate tracks

1 From the pop-up menu, choose Select By, then choose Layer.

2 Select the layer containing tracks for TOP.

3 From the tool palette at the left edge of the window, choose the Convert to MAX tool.

4 From the upper drop-down list, select Tracks.

5 From the lower drop-down list, select TOP.

6 Make sure the Delete selected entities after conversion option is selected, then choose the OK button.

7 Repeat steps 1 through 6 for the bottom layer and any inner copper layers, selecting BOTTOM and INNERn at step 5.

To complete the translation

When you finish, exit Visual CADD but do not save the modified DXF file.

Note: For any track that ends inside a pad, but slightly off center, a small end segment is added to the track to carry it to the center of the pad.

The translation process completes, and Notepad displays any warning or error messages generated.

As your tracks are imported, nets are assigned the names N00001, N00002, and so on. As tracks that begin on one layer join with tracks on other layers through vias, some of the created netnames are no longer necessary and are eliminated. This causes the "gaps" in the net naming sequence that you may notice in the translated board file.

Stopping and resuming an incomplete translation

The process described in this lesson can be time-consuming, and you may need to stop the translation temporarily and complete it later.

To pause during translation

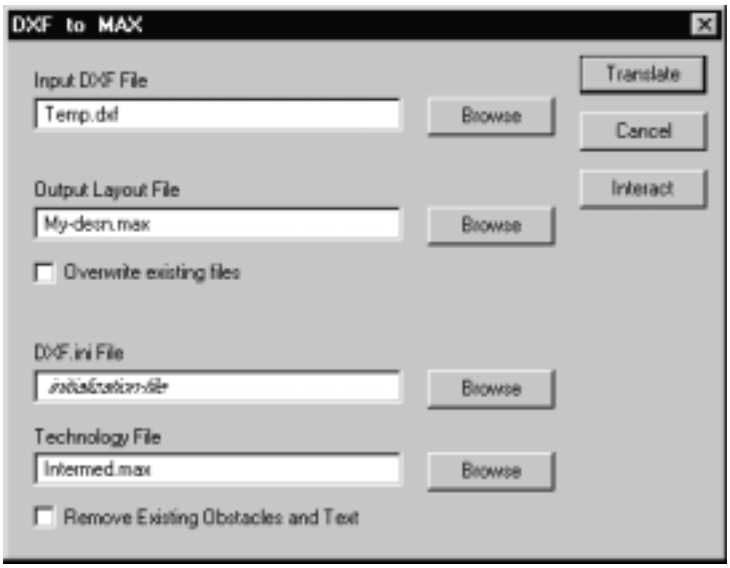
1. Save the DXF file you're working from (MY-DESN.DXF, for this example), using a name like TEMP.DXF.
2. Rename the MAX file you've created so far (MY-DESN.MAX), using a name like INTERMED.MAX. This will be the technology file when you resume the translation.

   Note: It is rarely advisable to use the same name for the output Layout file and the technology file (skipping step 2). The DXF translation is irreversible, and a system failure or other problem could corrupt all the work you had done before pausing, as well as after resuming. By using a different name for the output Layout file in the next procedure, your loss would be limited to work done after resuming the translation.

To resume the translation process

1. Open the Layout session frame.

2 From the File menu, choose Import, then choose DXF to MAX.



3 In the Input DXF File text box, type the name of the temporary DXF file (from step 1 of the preceding process).

4 In the Output Layout File text box, type the name of the MAX file you were creating when you paused.

5 In the Technology File text box, type the name of the intermediate MAX file (from step 2 of the preceding process).

6 Make sure the Remove Existing Obstacles and Text option is not selected. Selecting this option will destroy your previous work.

7 Choose the Interact button. The screen clears, then Visual CADD opens displaying the input DXF file.

8 Continue the translation where you left off.

Importing a design from multiple DXF files

If the printed circuit board exists in DXF format spread across several files, your work follows the same general flow: translate the components first, then the tracks. Each DXF file requires a separate execution of the DXF-to-MAX translator, and you should use a separate intermediate MAX file for each DXF file.

If the pad definitions are in the same DXF file as tracks, but the silkscreen elements are in a separate DXF file, you can postpone component silkscreen translation until after the copper layers are translated.

As long as the component pads are translated before the tracks are translated, you are free to reorder the rest of the process to suit the manner in which your board file has been split across DXF files.

To import a multiple-file DXF design

1. Translate the first DXF file, using these settings:
   - ❑ Output Layout File: INTRMED1.MAX
   - ❑ Technology File: DEFAULT.MAX
2. Translate the second DXF file, using these settings:
   - ❑ Output Layout File: INTRMED2.MAX
   - ❑ Technology File: INTRMED1.MAX
3. Translate the third DXF file, using these settings:
   - ❑ Output Layout File: INTRMED3.MAX
   - ❑ Technology File: INTRMED2.MAX
4. Continue in this manner, using the output file from one translation as the technology file for the next, until you have processed all of the DXF files.

   The last file cited in the Output Layout File text box (INTRMEDn.MAX) contains the completed design.

GerbTool

17

GerbTool is a powerful tool for viewing and modifying Gerber files and running netlist comparisons.

GerbTool can be used for many purposes in the Layout design flow. A few of the uses include:

- **Gerber verification** - verify that Gerber files contain accurate data.
- **Gerber to IPC-356 netlist comparison** - generate a new netlist in GerbTool, and compare that netlist with an IPC-356 netlist created from the Layout design file.
- **Panelize** - create multiple copies of a design in one film box. This allows multiple copies of a design to be manufactured as one panel.
- **Teardrop pads** - adds copper to trace entries into pads and at T junctions to ensure connectivity.

The GerbTool Design File

You can quickly open and examine all of the Gerber files for a design by opening the GerbTool Design file (.GTD) that Layout generates during post processing.

1 In Layout, from the Auto menu, choose Run Post Processor.

2 In the Layout session window, from the Tools menu, point to GerbTool and choose Open.

3 In the Open dialog box, navigate to the GTD file created in step 1 and click OK. GerbTool launches, imports the Gerber files, and displays the Gerber files. To view the import report, click Yes in the Import Gerber dialog box. The import report file lists all files that were converted and

4 Use the Color Bar to view or hide layers. Hover over each layer number to view the Gerber filename for that layer in a tooltip. When a Layer number has a red box around the number this layer is currently visible and editable. Layers without red boxes are not visible or editable. A black box around the layer number indicates that the layer is visible for reference only and not editable.

Gerber to IPC-356 netlist comparison

You can compare an IPC-356 netlist with a GerbTool netlist to detect opens and shorts that Design Rule Check and Statistics in Layout cannot detect. You create the IPC-356 netlist and a .GTD file using Layout. You create the GerbTool netlist and run the comparison in GerbTool. GerbTool creates a report that lists any discrepancies found.

Mandatory Conditions

The GerbTool IPC-356 netlist comparison will only work if the following conditions are met in Layout:

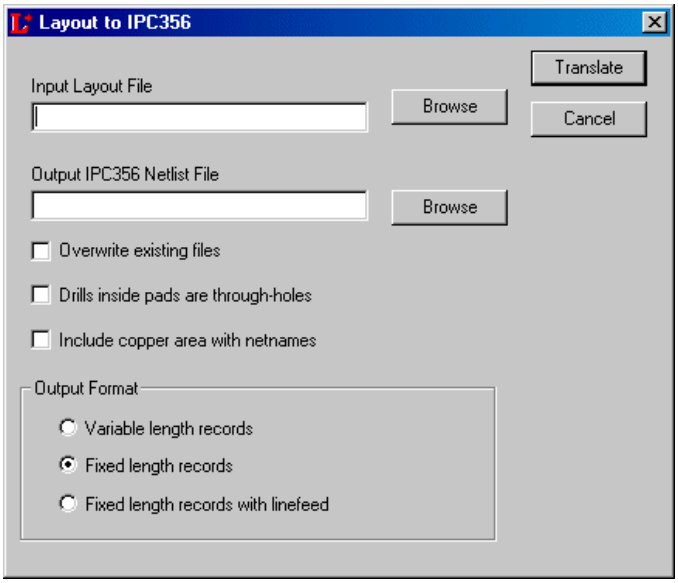
- The board must be 100% routed, with no partials routes or unroutes.
- There must be no errors reported when Route Spacing Violations, Copper Continuity Violations and Check Copper Pour are checked, using Design Rule Check.
- The Flood Planes/Pours option cannot be assigned on a Plane Layer of any padstack.
- The board cannot have any padstacks defined as blind or buried.

Creating the Gerber files in Layout

1. Save the .MAX file to a new, empty, temporary location.
2. From the Options menu, Choose Post Process Settings. The Post Process spreadsheet appears.
3. Select all etch layers (Routing and Plane) and select Properties from the pop-up menu. The Post Process Settings dialog appears.
4. Check the Enable for Post Processing option and click OK.
5. Select all of the layers that are Batch Enabled and from the pop-up menu, choose Properties.
6. In the Post Process Settings dialog box:
   a. Clear the Center on page and Mirror check boxes.
   b. Ensure the X Shift and Y Shift values are both zero. This ensures that the coordinates in the IPC-356 netlist match the coordinates in the Gerber files.
   c. In the Format group box, choose RS-274D. Do not use Extended Gerber.
   d. Click OK.
7. Left click the spreadsheet and from the pop-up menu, choose Run Batch to create the Gerber files and .GTD file. No thruhole.tap file is required.
8. Close the Post Process spreadsheet.
9. Save and close the .MAX file.

Creating the IPC-356 netlist in Layout

1 In the Layout session window, from the File menu, point to Export and choose Layout to IPC-356.



2 In the Input Layout File box, browse to the .MAX filename.

3 The file name in the Output File box will have a .NET extension. Change it to an .IPC extension. This makes it is easier to find in GerbTool.

4 In the Output format group, check the Fixed length records option. No other options need to be checked.

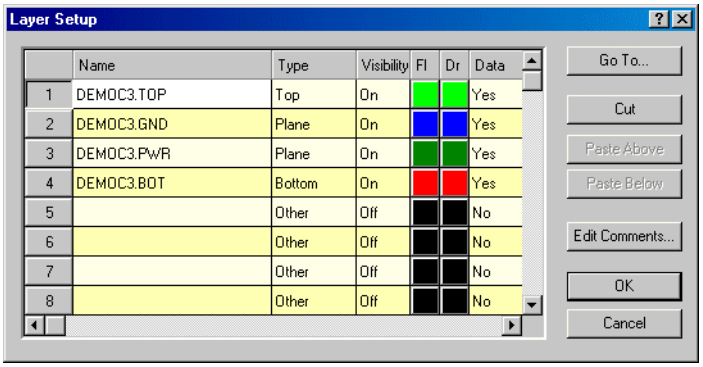
5 Click the Translate button to create the netlist.

Running the netlist comparison in GerbTool

1 In the Layout session window, from the Tools menu, point to GerbTool and choose Open.

2 Navigate to the .GTD file and click Open.

3 From the Setup menu, choose Layers.



4 In each layer, click the Type cell and select the correct layer type in the drop down box. Use Top, Bottom, Inner or Plane. Click OK when finished.

Note: If you changed a layer type in Layout it will come into GerbTool as the original type. You must set it to the new, correct type in GerbTool. For example: If you changed an Inner Layer to a Plane Layer in Layout, it will come into GerbTool as an Inner Layer. You must change it to Plane Layer in GerbTool.

5 Ensure that the top layer is the first layer in the list, and the bottom layer is the last layer. Use the Cut and Paste Below or Paste Above commands to change the order of the layers.

6 Left-click inside the name field of the next available layer entry below the Bottom layer. The data column should say "No" which indicates that there is no data on this layer yet.

7 Name the layer “Drill” and set the Layer type to “Drill”.

8 Left-click inside the name field of the next available layer entry below the Drill layer. The data column should say "No" which indicates that there is no data on this layer yet.

9 Name the layer IPC and keep the Layer type set to "Other".

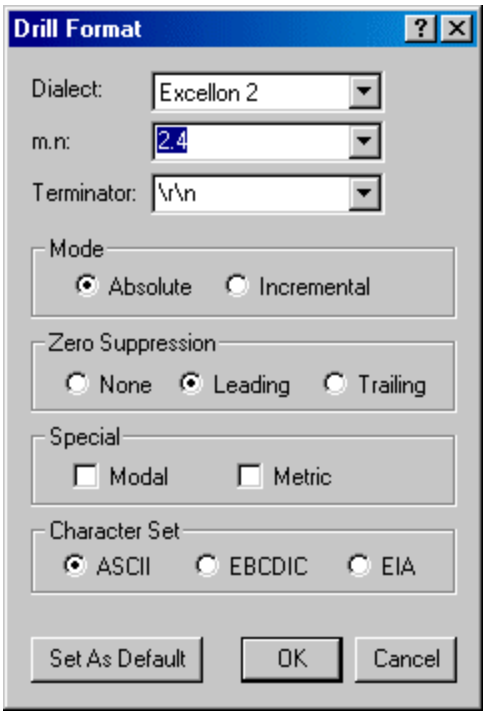
10 Click OK.

11 Select the Drill layer as the active layer from the drop-down layer list.

12 From the File menu, point to Import and choose Drill. The Import Drill dialog appears.

13 From the Files of Type drop down box, choose All Files (*.*).

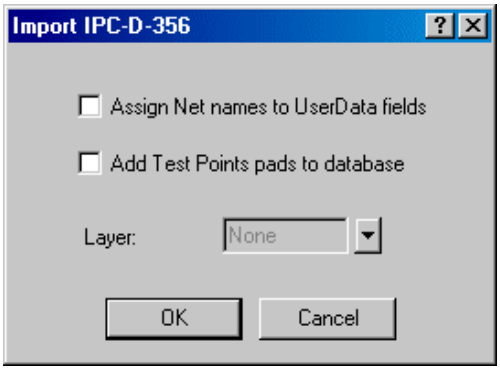
14 Navigate to the THRUHOLE.TAP file created by Post Processing in Layout and click Open. The Drill Format dialog box appears.



15 Accept the default values and click OK.

16 Select the Drill layer as the active layer from the drop-down layer list.

17 From the File menu, point to Import and choose IPC-D-356. Select the .IPC file in the Open dialog and click OK.



18 In the Import IPC-D-356 dialog box, uncheck both options and click OK.

19 You will see a message:

This message refers to the netlist that GerbTool generates to compare against the IPC-356 netlist.

20 Click Yes and GerbTool creates the new netlist.

21 Click OK in the dialog box where GerbTool indicates the number of errors found.

22 Click Yes in the dialog box where GerbTool asks if you want to view the errors.

23 A list of the errors is opened in Notepad. Use the following table to interpret the errors.

Error	Explanation
IPC Net Re-assignment, GerbTool nets 156 192 Locations: 0.7835, 0.1875 and 1.5791, 0.1850 IPC net PLLVEE:L3(1)	This is an open detected in the IPC-D-356 netlist. The items at the two locations are listed as belonging to the PLLVEE net in the IPC netlist, but GerbTool's internally generated netlist based on the gerber data show no connectivity between the items in the artwork.
Gerber Net Re-assignment: GerbTool net 2409 Locations: -0.0440, -1.5500 and 2.0050, 2.4450 IPC nets GND:~FV102(FV) DIFFSNS:P1(16)	This is a short detected in the IPC-D-356 netlist. The items listed ~FV102 and P1.16 are listed in the IPC netlist as belonging to two different nets: GND and DIFFSNS. Yet, GerbTool's internally generated netlist (based on the gerber data) has found connectivity between the two items and has assigned them to net number 2409 in GerbTool's internal database.
No IPC data for location 0.7920, 0.0562 Layer:2 No Gerber data for location 0.0569, 0.5581 ID N/C:J3(M1) idx 0	These errors usually appear if the bottom etch layer has not been moved so that it is the last etch layer listed among the gerber layers loaded into the current design.
No Gerber data for location 1.5980, 4.3800 ID 45:() idx 43	There was an IPC-D-356 record for this location, but no Gerber data.

Error	Explanation
No IPC data for location 2.8750, 3.7500 Layer:1	There is a pad on this layer that does not have any matching IPC information.

Panelize

The Panelize command creates multiple copies of a design in one film box. This allows multiple copies of the design to be manufactured as one panel. When you choose the Panelize command, GerbTool displays the Panelize/Vent Parameters dialog box.

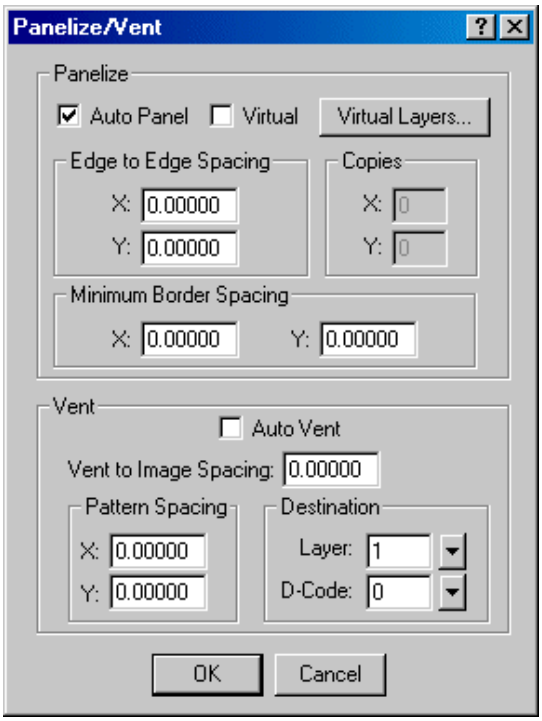


Figure 17-1 Panelize/Vent dialog box.

Auto Panel Provides automatic placement of the maximum number of copies of the board image, on a single panel.

Virtual Allows GerbTool to panelize your design without actually duplicating layer data in the database. Virtual

panelization provides many benefits, including automatic updating of all images during edits and drastically reduced file sizes. Further, if you want to plot your design on an extended Gerber or FIRE9xxx compatible plotter, GerbTool automatically inserts the proper step-and-repeat codes into your Gerber data.

Edge to Edge Spacing (available when Auto Panel is selected) The values set the minimum distance between adjoining edges of the image copies in the X and Y axes.

Point to Point Spacing (available when Auto Panel is not selected) The values set the minimum distance between adjoining edges of the image copies in the X and Y axes.

Copies The values set the number of copies in the X and Y axes.

Note: Although GerbTool copies only visible layers, all layers of the original image remain aligned after panelization.

Minimum Border Spacing Values set the minimum distance between the edges of the image copies and the edge of the panel.

Auto Vent Defines the shape and placement of flashes on the panel outside the image areas. GerbTool adds the pattern spacing and aperture selection to the database. Automatic venting can occur during panelization, regardless of whether or not the panelization is automatic. Venting may be targeted to any layers in the Gerber file.

Vent to Image Spacing Sets the spacing between the image copies and the venting area.

Pattern Spacing Sets the spacing between the flashes in the vent pattern.

D-Code Sets the size and shape of the flashes.

Manual panelization

To perform manual panelization

1. Activate those layers you want to include in the panelization.
2. Select the Panelize command from the Tools menu. GerbTool displays the Panelize/Vent dialog box.
3. Uncheck the Auto Panel option.
4. Enter the number of rows and columns in the Copy fields.
5. Enter the Edge to Edge and Minimum Border Spacing values.
6. Choose the OK button.
7. Draw a selection box around the area you want to copy.

GerbTool previews the panel layout. After asking for confirmation, GerbTool completes the panelization process. Depending on the setting of the Virtual button, GerbTool either copies the proper number of images into the database or notes the number of copies and their location for display purposes.

Automatic panelization

To perform automatic panelization

1. Activate those layers you want to include in the panelization.
2. Select the Panelize command from the Tools menu. GerbTool displays the Panelize/Vent dialog box.
3. Check the Auto Panel option.
4. Enter the Edge to Edge and Minimum Border Spacing values.
5. Choose the OK button.

GerbTool automatically calculates the maximum number of images that fit inside the current film box, then previews the panel layout. After asking for confirmation, GerbTool completes the panelization process. Depending on the setting of the Virtual button, GerbTool either copies the proper number of images into the database or notes the number of copies and their location for display purposes.

Automatic venting

To perform automatic venting

1. Check the Auto Vent button within the Panelize editing dialog box.
2. Enter the Vent to Image Spacing values and the Pattern Spacing values.
3. Enter the D-Code values. After panelization, GerbTool fills the specified area with the defined pattern of flashes.

Note: In both automatic and manual venting, the style of vent pattern is customized using custom apertures. For example, you can create a hatch or cross hatch pattern using a diagonal or cross shape custom aperture. Be sure to set the height and width of the overall size of the custom aperture in the aperture list.

Virtual panelization

To perform virtual panelization

1. Activate those layers you want to include in the panelization.
2. Select the Panelize command from the Tools menu. GerbTool displays the Panelization editing dialog box.
3. Select the Virtual Layers button. GerbTool displays a dialog box listing the loaded layers.
4. Select those layers you want to include in the virtual panelization and choose the OK button.

5 Proceed with either automatic or manual panelization.

GerbTool modifies the database with the number of copies and their location for display purposes. Virtual panel mode (and hence the display of virtual panels) may be toggled on or off using the CTRL+V shortcut key.

GerbTool also inserts step and repeat codes into NC Drill output data if you select the Virtual button. This may be necessary to drill large panels if your NC equipment is memory limited.

Note: Although no data is duplicated during virtual panelization, the data origin is modified to center the images within the panel. Therefore, it is still necessary to save your design after panelization.
If your designs are plotted on a plotter that does not support step-and-repeat codes, you must run the Panelize command without the Virtual button selected and save your panelized Gerber files before you send them to the plotter.

Teardrop Pads

Creating teardrop pads can help ensure connectivity by adding copper to the trace entry points of pads and T-joints.

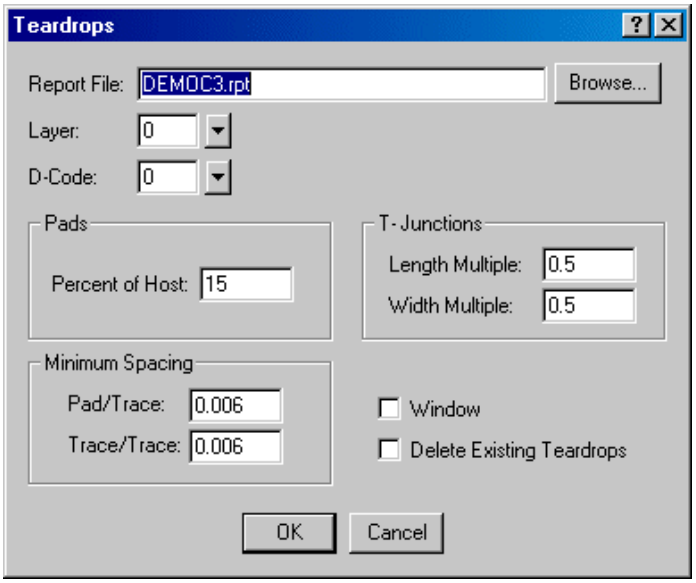
To create teardrops on pads and t-junctions

1 Open the GerbTool Design (GTD) File.

2 In the color panel, toggle on layers that you wish to teardrop. Layers that are toggled on are indicated by a red box around the layer number. Only these layers will be affected by the teardrop modifications.

3 Click the redraw (eye glasses) button to redraw the screen. The only layers now visible are the layers you wish to modify.

4 From the Tools menu, choose Teardrops. The Teardrops dialog box appears.

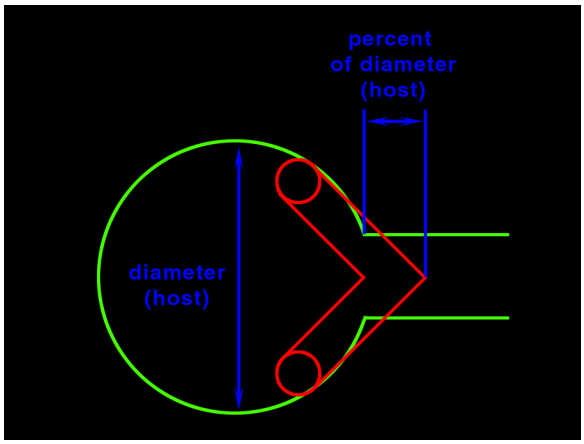


5 Enter a report file name

6 In the Layer box, choose 0 to apply the teardrops to all visible layers. To limit teardrops to a specific layer, choose that layer from the drop-down box.

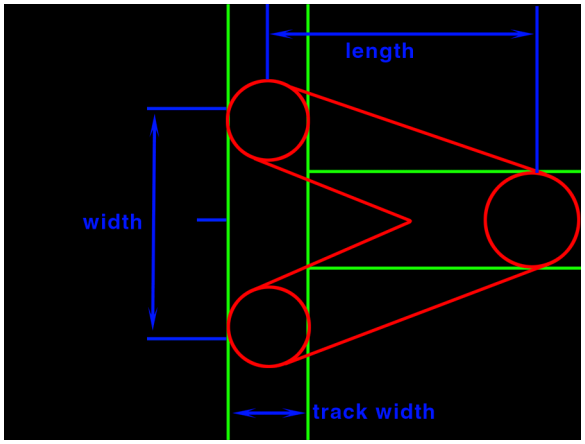
7 In the D-Code box, choose 0 to apply teardrops to all D-Codes. You can limit teardrops to only elements that use a specific D-Code by selecting it from the drop-down box.

8 The Percent of Host box in the Pads group box sets the size of the teardrop relative to the diameter of the pad. Values greater or less than 100% are acceptable.



Set an appropriate percentage in the Percent of Host box.

9 Settings in the T-Junctions group box indicate how teardrops will be drawn relative to the width of the track.



The width and length of the tear drop is expressed in a multiple of the width of the track. Type in appropriate values in the T-Junctions group box.

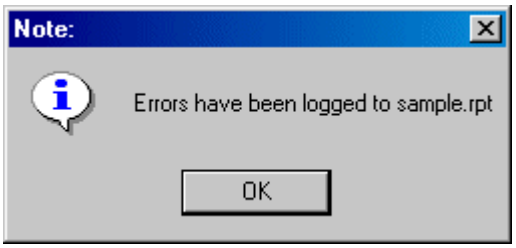
10 The Minimum Spacing group box applies to the Design Rules Check (DRC) for the distance between a teardrop and a line or another pad. Type in appropriate values for the Pad/Trace and Trace/Trace settings.

11 Click OK to create the teardrops.

Note: If you would like to select a specific area of the design on which to create teardrops, select the Window option. Click OK. In the GerbTool design window, click and drag to select the area in which to create teardrops.

12 When prompted, click Yes to generate a netlist. This generates an internal netlist in GerbTool from the current design.

13 When teardrop creation is complete, you will see the following message:



14 Click OK to continue.

15 When prompted, click Yes to view report. The report is displayed with Notepad. The report lists the number of teardrops added per layer, and the locations where GerbTool was unable to add teardrops.

16 In the GerbTool design window, pads on which GerbTool couldn't add teardrops are highlighted white. To more easily see the errors, zoom in on the error, and from the View menu choose Sketch.

To delete existing teardrops

1 From the Tools menu select Teardrops. The Teardrops dialog box appears.

2 Check the Delete Existing Teardrops option.

3 From the Layer drop down box, choose the layer from which to remove teardrops. To select all visible layers, choose layer 0.

4 Click OK to remove the teardrops.

Dialog box descriptions

18

The following is an exhaustive set of descriptions for the dialog boxes you may encounter while using OrCAD Layout. Each description is listed alphabetically, using the dialog box title.

Add Color Rule dialog box

To display the Add Color Rule dialog box, choose the Color Rules command from the Options menu.

Rule

The options in this settings group specify which type of entity or object is affected by the new color rule that you intend to create.

Layer

The value you enter in this text box determines the layer that the new color rule affects. You can either enter the layer nickname or the layer number (as shown in the layer drop-down list of the design window). You can also use a dash (-) to specify any or all layers; this applies the color rule to any occurrence of the object you selected in the Rule settings group (above).

Related topics

Color dialog box

Colors command

Color Rules command

Add Component dialog box

The Add Component dialog box appears when you choose New from the pop-up menu with the Component tool active and no components selected.

Reference Designator

In this text box you specify the reference designator you want to assign to the new component.

Part Type

Enter a part type number for the new component in this text box; by doing so, you'll be able to associate components of the same type for purposes of placement.

Value

Use this text box to assign a value to the new component. For discrete components, values are typically associated with electrical properties.

Footprint

Choosing this button displays the Select Footprint dialog box, which allows you to assign a footprint to the new component.

Location

X and Y location

These two text boxes permit you to designate the location where the new component appears on your board. The location of the design datum serves as the origin (0,0). The location of your cursor (when you choose the New command) provides default coordinates. If you change these values, the component appears at the coordinates you specified after you choose OK. Nevertheless, the component moves with your cursor and is not placed until you click the left mouse button.

Rotation

This text box allows you to indicate the initial arc of rotation for the new component. The new component rotates counter-clockwise.

Group Number

You can assign a group number to the new component by entering a value in this text box. Group numbers allow you to associate components of various types for placement purposes.

Cluster ID

Layout adds the new component to the cluster whose number you specify in this text box. If you want the new component to be the key component of a cluster, select the Key option (see below) and the cluster's name changes to the new component's reference designator.

Component flags

Fixed

This option permanently locks the new component to your board.

Not in Netlist

Selecting this option ensures that mechanical parts are not deleted during AutoECO.

Locked

When you select this option, autoplacement algorithms cannot adjust the position of the component.

Route Enabled

Selecting this option allows you to route connections into and out of the component.

Key

When you select this option and enter a value in the Cluster ID text box, Layout adds the new component to the specified cluster and designates this component as the key part within that cluster. The cluster's name also changes to the new component's reference designator.

Do Not Rename

Selecting this option prevents the Rename Components command from automatically renaming the component.

Related topics

Select Tool command (Component)

New command (Component)

Add Footprint dialog box

This dialog box appears as a result of choosing the New command on the pop-up menu while viewing the Footprints spreadsheet. In creating a new footprint, you do not place an entity on your design, but rather add a footprint to the Footprints spreadsheet.

Footprint Name

Enter a text string appropriate to the new footprint in this text box.

Pad Name

Use this text box to assign a name to a pad in the footprint you intend to create.

Pad X, Y

Specifies the pad's location on a footprint. The pad location is measured from the local datum.

Insert X, Y

Specifies the pad's location on a footprint. The pad location is measured from the local datum.

Padstack Name

When you add a footprint, you determine the size and shape of its pads by assigning a particular padstack to the footprint.

Any padstack listed in the Padstack Name drop-down list is available for this purpose. Those padstacks defined in the current design are labeled as “local.” Local padstack definitions appear in the Padstacks spreadsheet.

Pad Entry/Exit Rule

Standard

This rule specifies that the router must route out of the pad in the direction of the longer axis of oblong or rectangular pads unless they are attached to components of three pins or less, or unless they are at the ends of DIP packages. In these cases, the router is free to route out of the sides.

Any Direction

No pad entry or exit direction rules apply for this footprint.

Long End Only

For this footprint, the router must route out of the "long end" of the pads; that is to say, in the direction of the longer axis. There are no exceptions, in contrast to the Standard option (above).

Additional Rules

Allow via under pad

By default, through vias of the same net are not allowed under SMT pads. Select this option to override the default setting and allow vias under SMT pads. Vias belonging to a different net are never allowed under pads, unless they are buried vias.

Preferred Thermal Relief

This option ensures that Layout provides a thermal relief to the selected pad within a pad-to-pad connection. However, when such a pad connects to a via, the via is assigned a thermal relief, rather than the pad designated as preferred. If you select neither the via nor the pad as preferred for a thermal relief, then the via in this situation receives the thermal relief.

A preferred thermal relief is especially useful when you have a through-hole board and you need to make connections between a capacitor and an IC. If you designate the capacitor pad as preferred for a thermal relief, then Layout assigns it a the thermal relief.

Forced Thermal Relief

If you designate a pin as a forced thermal relief, a thermal relief is provided for that pin on pertinent plane layers. If you highlight the pad in a via/pad connection and select the Forced Thermal Relief option, both the via and the pad receive a thermal relief.

When you do not specify a preference, Layout always provides a thermal relief to the via in a situation involving a via and a pad.

Related topics

Edit Pad dialog box

Edit Padstack dialog box

Footprints spreadsheet

Thermal Relief Settings command

Add Free Via dialog box

The Add Free Via dialog box appears when you choose the Via and then the New command (Via) from the Tool menu. You

can create free vias using this dialog box; assigning nets, groups, and via types in the process. The Add Free Via dialog box also allows you to select and assign a footprint to a new free via. You can also lock a newly created via to your board with the Locked option.

Free Via *X (where X is an integer)*

The number of free vias you have created (but not the number present in your design) appears at the top of the dialog box. This number includes the free via currently being added.

Padstack Name

Select the name of a padstack type from the drop-down list. Free vias can only be assigned padstack types that are defined in the Padstacks spreadsheet.

Net Name

Free vias must be assigned to a net, regardless of their connectivity. Use this drop-down list to designate an associated net for a new free via.

Convert to Component

Choosing this button opens the Select Footprint dialog box. After selecting a library, choose a footprint for the free via you create.

Group Number

It’s possible to associate a free via with a component group while working in Layout (though it is recommended that you create groups at the schematic design level). Enter the group number you want assigned to the newly created free via.

Location

The text boxes in this settings group allow you to designate the X and the Y coordinates for the positioning of a newly created free via. If you leave these boxes blank and then choose OK, the free via you create moves with your pointer until you place it on your board with a click of the left mouse button.

Locked

This option locks a newly created free via in position after you place it on a board.

Related topics

New command (Via)

Free Via Matrix command (Place)

Select Footprint dialog box

Add Pad dialog box

This dialog box appears as a result of choosing New from the pop-up menu while a single pad is highlighted in the Footprints spreadsheet.

Pad Name

Each pad of a footprint has a unique name, which is used to keep track of connectivity. The connection list uses the component name from the component list, along with the pad.

Pad X, Y

Specifies the pad's location on a footprint. This can only be set when one pad is selected. The pad location is measured from the local footprint datum rather than from the board datum.

Padstack Name

When you are editing a footprint, you indirectly determine the sizes and shapes of the pads (the padstack) by assigning a particular padstack to the pad. Any padstack listed in the Padstack Name drop-down list can be assigned to a pad. The padstack definition is given in the Padstacks spreadsheet.

Pad Entry/Exit Rule

Standard

This rule specifies that the router must route out of the pad in the direction of the long axis of oblong or rectangular pads unless they are attached to components of three pins or less, or unless they are at the ends of DIP packages. In these cases, the router is free to route out of the sides.

Any Direction

No pad entry or exit direction rules apply for this footprint.

Long End Only

For this footprint, the router must route out of the "long end" of the pads.

Additional Rules

Allow via under pad

By default, through vias of the same nets are not allowed under SMT pads. If you want to override this limitation, you may do so by setting this flag to Yes. Vias belonging to a different net are never allowed under pads, unless they are buried vias that will not create a short.

Preferred Thermal Relief

With this option, you can ensure that the newly created pad is assigned a thermal relief when a single wire connects two pads. The exception is when a preferred thermal relief pad is connected by a single wire to a via; in which case the via is assigned the thermal relief.

A preferred thermal relief is especially useful when you have a through-hole board and you need to make connections between a capacitor and an IC. If you designate the capacitor pad as preferred thermal relief, then the appropriate pad of the pad-to-pad connection is assigned the thermal relief.

Forced Thermal Relief

If you designate any pin of a footprint as a forced thermal relief pad, the pin receives a thermal relief on the plane layers attached to the relevant net.

Related topics

Edit Footprint dialog box

Edit Padstack dialog box

Footprints spreadsheet

Thermal Relief Settings command

Add Test Point dialog box

The Add Test Point dialog box appears when you choose the New command (Test Point) after choosing Test Point from the Tool menu. Using this dialog box, you can create test points and in the process, assign them to nets, groups, and give them via types. The Add Test point dialog box also allows you to select and assign a footprint to a new test point. You can also lock a newly created via to your board with the Locked option.

Test point *X (where X is an integer)*

The number of test points you have created (but not the number present in your design) appears at the top of the dialog box. This number includes the test point currently being added.

Padstack Name

Select the name of a padstack type from the drop down list. Test points can only be assigned to padstacks that are both defined and enabled for use as a test point.

Net Name

Use this drop-down list to designate an associated net for a new test point.

Convert to Component

Choosing this button opens the Select Footprint dialog box. After selecting a library, choose a footprint for the test point you create.

Group Number

It's possible to associate a test point with a component group while working in Layout. Enter the group number you want to assign to the newly created test point.

Location

The text boxes in this settings group allow you to designate the X and the Y coordinates for the positioning of a newly created test point.

Locked

This option locks a newly created test point in position after you place it on a board.

Related topics

Edit Padstack dialog box

Properties command (Padstack)

Padstacks spreadsheet

Advanced Options dialog box

The Advanced Options dialog box appears when you select the Advanced Options button in the Stackup Editor dialog box.

The options in this dialog box specify the overall properties of the board.

Desired Board thickness

Specify the desired thickness of the completed PCB along with the tolerance value.

Desired Impedance of the board
Specify the desired impedance of the board.

Adjust dielectric for the desired board size.

Select this button to adjust the thickness of the dielectric material used between the layers, such that the total board thickness is with in the tolerance range specified by you in the Desired Board Thickness text box.

Assign Via dialog box

To display the Assign Via dialog box, choose the Nets command from the spreadsheet toolbar button menu. The Nets spreadsheet appears. Select one or more cells and then

choose the Assign Via per Net command from the pop-up menu.

Via*N* (where *N* is an integer)

In order to use a specific via for the routing of a particular net, select the net in the nets spreadsheet, choose the Assign Via per Net command from the pop-up menu and then select an available via. After you choose OK, the via you selected is assigned to the net you originally highlighted in the Nets spreadsheet. A via is present in the Assign Via dialog box only after it has been defined in the Edit Padstack dialog box.

Selecting a via for a particular net does not prohibit any other net from using that via. The assignments made with this dialog box override the Use All Via Types option in the Route Settings dialog box. You can therefore check the Use All Via Types option and still specify certain vias for certain nets by assigning them with the Assign Via dialog box.

If you do not check the Use All Via Types option, you must specifically assign vias to nets that need to be routed with a specific via type. Otherwise, the router will use the default via (Via 1).

Related topics

Edit Net dialog box

Edit Padstack dialog box

Autodimension Options dialog box

The Autodimension Options dialog box contains options for creating dimension markings. To display the Autodimension Options dialog box, choose the Properties command (Dimension) command from the Tool menu. If the Properties command is unavailable, choose Select Tool. Then, with the pointer in the design window, press INSERT and choose Properties from the pop-up menu.

Dimension Type

Relative Dimensions

When you select this option, Layout creates dimensions between two end points. The dimension is measured from the starting point (the first point at which you click the left mouse button) to the termination point (the second point at which you click the left mouse button).

Absolute Dimensions

When you select this option, Layout creates dimensions based on the datum. That is, the dimension is measured from the originating datum to the point at which you click the left mouse button.

Arrow Style

Open Arrow

When you select this option, Layout creates an arrow using two lines to illustrate the dimension.

Solid Arrow

When you select this option, Layout creates a “solid-fill” arrow to illustrate the dimension.

Line Width

This option sets line width for dimension marks.

Text Height

This option sets text size for a dimension.

Layer

Layout places the dimension on the layer you choose from the drop-down list.

Related topics

Move Datum command

Properties command (Dimension)

New command (Dimension)

Backup Settings dialog box

In the Backup Settings dialog box, you can set the time between design backups and specify the number of backup files you want Layout to store. To display this dialog box, choose the Auto Backup command from the Options menu.

Layout automatically places backup files in your working directory. The number of backup files that Layout creates is dependent upon the integer you enter into the Number of backups to keep text box (see below). When this number of files is created, Layout overwrites the oldest backup file with the second oldest when it creates a new backup.

Backup time (in minutes)

Indicates the interval between automatic backups of the current board design file.

Number of backups to keep

The integer you enter here indicates the number of backup files that Layout creates in your working directory--or the directory specified in the Directory for backups text box (see below).

Backup after each sweep

This option tells Layout to create a backup file between routing sweeps. By default, Layout places this file in the current working directory. Its name matches the sweep that you just completed.

Directory for backups

Specifies the directory for storage of BACKUP*.MAX. If you are running Layout from a network, you may want to store the backup files locally.

Related topics

Save command (File menu)

Save As command (File menu)

Backup command (File menu)

Check Design Rules dialog box

In the Check Design Rules dialog box, you can specify which design parameters you use to verify the layout of a board. After selecting any number of the Check Rule settings (below), choose OK. Layout conducts a design rule check and any problems are marked by a circle and can be queried by using the Select Tool command (Error), selecting the error, then choosing the Query Window command (View menu).

Check Rule Settings

Placement Spacing Violations

Selecting this option verifies component-to-component spacing parameters, such as height restrictions, insertion outline spacing and grid restriction violations. Each component has an outline that encompasses the entire area

of the IC or discrete component, including such objects as pin-out patterns and sockets. Layout uses this outline to determine possible violations.

Route Spacing Violations

The Route Spacing Violations setting verifies adherence to route spacing specifications, as defined in the Route Spacing spreadsheet. Layout uses the information on this spreadsheet to autoroute, maintain the integrity of the DRC-enabled environment and conduct design rules checks when you request them.

Net Rule Violations

When you select this option, the design rule check you conduct will ascertain whether any nets violate the parameters appearing on the Nets spreadsheet.

Copper Continuity Violations

This setting instructs Layout to check for copper that is either associated incorrectly (for example, a heat sink connecting to the wrong net) or entirely unassociated (for example, a heat sink that has no connectivity).

Via Location Violations

With this option selected, the design rules check you conduct verifies that the location and spacing of all vias corresponds to specified parameters.

Off Grid Vias

With this option selected, the design rules check looks for off-grid vias.

Pad Exit Violations

This option pertains to pad shape.

SMD Fanout Violations

Selecting this option allows for a design rules check on any enabled nets (as indicated in the Nets spreadsheet) that originate at SMD pads and end at neither a through hole nor a via.

Test Point Violations

By selecting this option, you instruct Layout to verify that every net that is enabled for a test point actually contains a test point.

Check Copper Pour

When this option is checked, Layout checks for spacing violations in copper pour. If you specify a clearance value when you define the copper pour, Layout uses the clearance value. Specify the clearance value in the Edit Obstacle dialog box (Copper Pour Rules group) when defining the copper pour.

If you don't specify a clearance value, Layout checks for net spacing values. Define spacing for individual nets in the Nets spreadsheet (Net Spacing button in the Edit Net dialog box). If you don't specify net spacing values, Layout uses the values from the Global Spacing spreadsheet.

Check Detail Obstacles

A detail obstacle is an obstacle that is neither placed (in the sense that a component is placed) nor is it routed. It is usually used for silkscreens, drill information, and assembly drawings, and often attached to footprints.

Report DRC/Route Box Violations Only

When you select this option and then choose OK, Layout conducts a design rules check for only that area of a board that is encompassed by the DRC/Route box. Though this option limits the scope of a DRC, it hastens the completion of the check.

Do not delete Known Good DRC

Select this check box, when you do not want the DRC errors marked as know good to be deleted from the error list. When you select this check box, it might happen that you have fixed a particular error that was marked as known good, but the error still appears in the error list. To remove the fixed known good DRC errors, clear the check box and run the DRC.

Hide Known Good DRC

This check box is enabled only if the Do not delete Known Good DRC check box is selected. Select the Hide Known Good DRC check box if you want that the DRC errors marked as know good, should not appear on the Layout board as well as in the Error Marker spreadsheet.

Related topics

Implementing power and ground fanout

Circular Placement dialog box

You can use circular placement to position your parts in a circular pattern on a board. To display this dialog box, choose the Place command from the Auto menu and then choose Array command (Place).

In order to use circular placement, you must first choose the component. You can select a component directly on the board, or you can use the Footprint button to browse the footprint libraries and the list of the footprints presently on the board.

Footprint

Choosing this button displays the Select Footprint dialog box, where you can select a library and then a footprint for circular placement. If you do not select a component on the board, you need use this button to choose one. With this button, you have a choice of footprints local to the board and footprints in the footprint libraries.

Reference Des

The reference designator for added component. Default value shown is the next unused reference designator for the Layout design file. You may choose your own unique reference designator if you wish.

Group Number

Group number to associate with added components. The default is 0 (zero), meaning components do not belong to any group.

Circle Center X, Y

Coordinates of the circle center. These are calculated from the board datum. When you establish this value, the board datum is temporarily moved to the circle center, and is restored to its original location once the circular placement is complete.

Circle Radius

Radius of the circle of placed components. The radius is measured from the circle center to the component placement point.

Start Angle

The angular location of the first component placed. Zero degrees is equivalent to three o'clock. If a component has been selected on the board, changing this value will cause the component to move and placement to begin from the new location.

Rel Start X, Y

Coordinates, relative to the circle center, for the first component added. If a component has been selected on the board, changing these values will cause the selected component to move and placement to begin from this new location.

Place Comp By

Use this drop-down list to establish how the component is placed on the circle circumference. The choices are Datum (the default), Center, Pin 1, or Insert Pt (insertion point).

Note: Regardless of how the component is placed, if it is subsequently rotated, it rotates about the datum.

Comp Count

The total number of components to be added, inclusive of a selected component.

Use Angle to Fill

Radio button to select between Angle to Fill and Angle Between

Angle to Fill

The portion of the circle to be filled by the added components.

Use Angle Between

Radio button to select between Angle to Fill and Angle Between

Angle Between

The space or angle between neighboring components.

Comp Angle

The rotational angle of each added component. Changing this will cause a selected component to rotate.

Comp Angle Increment

A successive rotation angle increment for each added component, calculated before placement. This will not affect a selected component, however each added component will be rotated by this increment relative to the previous component. For example, starting at 0, a Component Angle Increment value of 20 degrees would cause components to be rotated before placement at 0, 20, 40, 60, etc. degrees.

Added Comp Angle

A rotational angle to be added to each component after it is placed, and therefore an angle about the component's placement point. For example, starting at 0, if Comp Angle Increment is 20 and Added Comp Angle is 5 the component rotations would be 5, 25, 45, 65.

Note: Changing Circle Center X,Y will cause Circle Radius, Start Angle, Rel Start X,Y, and Comp Angle to be updated automatically.

When you set the Start Angle, the Comp Angle is automatically set to the same value; in most cases these two options should have the same value.

Note: Changing Comp Count will auto-update Angle to Fill, Angle Between, and Comp Angle Increment. Setting Angle Between will automatically-set Comp Angle Increment to the same value, but setting Comp Angle Increment will not change anything else. Changing Angle to Fill will automatically-update Angle Between and Comp Angle Increment. In most cases, Angle Between and Comp Angle Increment should be the same, however, you can change Comp Angle Increment and it will not update any other value.

Circular tab (Pad Array Generator)

Use this tab to create Circular pad arrays.

X Direction area

Number (p) Total number of pads in the array.

Spacing (x) Disabled

Start Value The value for the first pad in the array. The start value must be entered in the entry box as a number, and is converted to numeric or alphabetic pad labels. Zero corresponds to the letter “A” when using alphabetic pad labels.

Increment Increment at which pads are labeled. For example, if you wanted to skip from 1 to 3 to 5 and so on, you would enter a value of 2.

Numeric or Alphabetic Sets alphabetic or numeric labeling. When using Alphabetic labeling, you can specify what letters to use in the Array Alphabet dialog.

Y Direction area

Number (q)	Disabled
Spacing (y)	Disabled
Start Value	Disabled
Increment	Disabled
Numeric or Alphabetic	Disabled

Options area

Radius (r)	Distance that the pads are from the center of the array, entered in the selected units. The radius cannot exceed 32 inches (812.8000 mm).
Angle (θ)	Angle in degrees between the current and next pad. The allowed range is negative 360° to 360°. Note that negative values change the Style Sample window display. **Note:** If you create more pads than can be located in 360° with the current angle setting, the Pad Array Generator will continue to place pads around the circle and possibly overlap pads.
Display Pad Name	Displays pad names in the Array Preview window. **Note:** Disabling this feature will speed up display refresh rates.
Display Drill	Displays relative drill information for each pad in the Array Preview window.

Silk Screen Allows you to set the silk screen spacing from the pin center.

The Notch check box is also provided in the Silk Screen group box. If you select this check box, the spacing will always be set wrt pin 1.

Place Outline Allows you to set the spacing of the outline from the pin center

Padstacks area

Default Padstack This sets the default padstack for all pads. Use the Select button to open the Select Padstacks dialog box, and select a padstack from the libraries.

Note: Changing the Default Padstack will always set Pin 1 to the Default padstack. To use a different Pin 1 padstack, set Pin 1 after you have set the default padstack.

Pin 1 Padstack Use this to override the Default Padstack for Pin 1. Use the Select button to open the Select Padstacks dialog box, and select a padstack from the libraries.

Note: You must set the Pin 1 padstack ***after*** you have set the default padstack.

Related topics

Pad Array Generator dialog box

Cleanup Design dialog box

The Cleanup Design dialog box enables you to specify what filters will be applied when you run Cleanup Design from the Auto menu.

Cleanup Routing

Miter 90 degree corners

This option miters all 90-degree corners.

Eliminate acute angles

This option removes all acute angle (angles of less that 90 degrees).

Optimize vertices

This option removes unnecessary corners wherever possible using a multiple pass algorithm that maximizes the probability of cleaning up situations where vertex removal must occur in a predetermined order.

Optimize shared tracks

This option eliminates situations where shared tracks (two tracks of the same net that overlap at some point) overlap, and ensures that the junctions between tracks don’t create acute angles.

Optimize shared vias

This option resolves any problems with vias of the same net overlap, preventing manufacturing problems.

Optimize pad exits

This option smooths and centers the pad exits for better manufacturing yield.

Override locked tracks

Normally, locked tracks will not be altered, but this option enables you to override the default behavior if you so desire. Note that routing disabled nets are never altered.

Cleanup Database

The following options help clean up unused objects in the database.

- Remove unused padstacks
- Remove unused footprints
- Remove unused nets

Color dialog box

In the Color dialog box, you can select colors for specific objects and board layers. To display the Color dialog box, choose the color toolbar button and double-click on any cell in the Color spreadsheet.

With the Color spreadsheet open, you can specify a different color for each type of entity in a design. Simply select New in the pop-up menu without selecting a table cell. This opens the Add Color Rule dialog box, which provides a number of options for the colored display of various board objects and entities, including layers.

If you wish to make an object invisible, select the appropriate row in the Color spreadsheet and then press the dash (-) key. Cells in the Color column of the spreadsheet that appear with pale diagonal stripes (rather than a solid block of color) indicate that the corresponding design object is invisible.

If an object is not present in the Color spreadsheet and you wish to make it invisible, select New from the pop-up menu, then select the appropriate option for the object type in the Add Color Rule dialog box and choose the OK button. In the Color spreadsheet, select the relevant row or cell and choose the Visible<>Invisible command from the pop-up menu.

Note: Invisible layers will be shoved by the autorouter. For example, if you are manually routing on the visible layers, and you insert a via, the router automatically clears and updates the invisible layers as well.

Note: If you are interactively routing, there is an easy way to bring your work to the fore of the design window: press the BACKSPACE key, followed by the number key corresponding to the layer you're routing. To revert to a view of your entire design, press the HOME key.

Related topics

Add Color Rule dialog box

Comp Attachment dialog box

The Comp Attachment dialog box appears when you choose the Comp Attachment button from the Text Edit dialog box. Using this dialog box, you can attach text to either a component or a footprint. Attaching text to a footprint causes all components that use that footprint to include the specified text.

Comp Attachment

- **None** - Specifies that no text is attached to the component.
- **Attach to Component** - Specifies that text is attached to the component that uses the footprint.
- **Reference designator** - Specifies the name of the component to which the text is attached.

Related topics

Component Attachment dialog box

Create New Footprint dialog box

Component Attachment dialog box

This dialog box appears when you choose the Comp Attachment button in the Edit Obstacle dialog box

In Layout, you can associate an obstacle with a specific pin, a specific component, or every component of a specific footprint. This association or attachment may or may not be electrical. For example, you may associate an obstacle with a pin or a place outline. In the first case, the attachment affects connectivity; the second situation, however, has no bearing on electrical operations.

You can also attach copper to a particular pin. In such a situation, the copper entity mimics the electrical attributes of that pin.

Component Attachment

Reference designator

Use this text box to specify the name of the component that you want to attach to the obstacle.

Pin Attachment

None

By selecting this option, you indicate that the obstacle is not associated with a particular pin of a component or footprint.

Attach to pin

By selecting this option, you attach the obstacle in question to a particular pin of a component or footprint.

Pin Name

Use this text box to indicate the name of the pin that you want to attach to the obstacle.

Related topics

Comp Attachment dialog box

Create New Footprint dialog box

New command (Obstacle)

Component Selection Criteria dialog box

The Component Selection Criteria dialog box appears when you choose Group, Select Filtered command (Component) from the Tool menu.

In this dialog box, you specify criteria for selecting a group of components. By default, Layout includes the surface mount and through-hole top and bottom component types, but excludes locked and fixed components from the selection process.

A layer must be visible for its components to be selectable, regardless of whether you are using single select, area select, or SHIFT + CLICK mode. If the top layer is invisible, top layer components are not selectable. The same is true for bottom layer components if the bottom layer is invisible.

Ref Des

If you leave this text box blank, Layout does not limit the selection process by component name. You can use the question mark wildcard (?) for one indeterminate character in the component name, or the asterisk wildcard (*) for a number of indeterminate characters in the component name. If you want to select a specific component, enter the name of the appropriate reference designator.

Footprint Name

If you leave this text box blank, Layout does not limit the selection process by footprint name. You can use the question mark wildcard (?) for one indeterminate character in the footprint name, or the asterisk wildcard (*) for a number of indeterminate characters in the footprint name. If you want to select a specific component, enter the name of the appropriate footprint.

Group Number

This option refers to the group number assigned to components at the schematic level. Assigning group numbers to components ensures that similar components are placed in close proximity. Leaving this text box blank does not limit the selection process to any group.

Component Types to Include

These options are selected by default. To filter any of the component types from selection activities, clear the appropriate check box.

Minimum Pins

Enter the minimum number of pins to serve as a criterion for component selection. Leaving this text box blank does not set a lower limit for the number of pins selected components possess.

Maximum Pins

Enter the maximum number of pins to serve as a criterion for component selection. Leaving this text box blank does not set an upper limit for the number of pins selected components possess.

Exclude placed

By selecting this option, you exclude placed components from the selection process.

Exclude locked

By selecting this option, you exclude locked components from the selection process. If you do not select this option, any locked components that meet your selection criteria have their locked flags cleared and become unlocked.

Exclude fixed

If you select this option, you exclude fixed components from the selection process. If you do not select this option, any fixed components that meet your selection criteria have their locked flags cleared and become unfixed.

Related topics

Components spreadsheet

Configure Design Library dialog box

The Configure Design Library dialog box appears when you choose Configure Design Library from the Tools menu in the Layout session window.

You can use this dialog box to create design libraries—libraries that contain all the footprints required for a design. For more information, see Creating design libraries on page 570.

Input MNL netlist file

Enter the path and filename of the netlist (.MNL) file for the design or click the Browse button to select the file.

Output library file name

Enter the path and filename for the design library you want to create.

Units

Select the unit (English or Metric) in which you want the footprints to be saved in the design library.

Search Criteria

Footprint Name

Select this check box if you want find only the footprints that have the same name as the footprint specified for the part in your OrCAD Capture schematic (using the PCB Footprint column in the property editor in Capture), or where the footprint is not specified, the part name in your Capture schematic.

Number of Pins

Select this check box if you want to find only the footprints that have an equal or greater number of pins as the part in your OrCAD Capture schematic and also have the same pin names as the pin names on the part in your Capture schematic.

List of libraries

Displays the list of libraries that will be searched for finding footprints.

Add Configured Libraries

Select this check box if you want to search the libraries you have made available for use in Layout. For more information

on making a library available for use in Layout, see Making libraries available for use on page 531.

The libraries you have made available for use in Layout are displayed in the list.

Add to List

Click this button to add libraries you want to search.

Note: Adding libraries to the list does not mean that the libraries are added to the list of libraries made available for use in Layout. You add libraries in the list only for specifying the libraries you want to search for footprints. For more information on making libraries available for use in Layout, see Making libraries available for use on page 531.

Remove

Click this button to remove a library from the search list.

Note: Removing libraries from the list does not mean that the libraries are removed from the list of libraries made available for use in Layout. You remove libraries from the list only to exclude them libraries you want to search for footprints. For more information on making libraries available for use in Layout, see Making libraries available for use on page 531.

Remove All

Click this button to remove all the libraries from the search list.

Search

Click this button to search for footprints to be added to the design library.

For more information on how Layout searches for the footprints to be added to the design library, see How Layout selects footprints to be added to a design library on page 574.

Search Result List

Symbol/Capture Footprint

Displays one of the following:

- The footprint name that is specified for a part in your OrCAD Capture schematic (using the PCB Footprint column in the property editor in Capture).
- The part name, if a footprint name is not specified for a part in your OrCAD Capture schematic.

Footprint

Displays the name of the footprint corresponding to each part in your OrCAD Capture schematic.

Replace Selected Footprint

Lets you change the footprint selected for a part.

Select the row for the part and click this button. The Replace Footprint dialog box appears. Perform a search for the footprint, select the footprint in the Footprint list and click Replace.

Save

Click this button to save the selected footprints to the design library.

Before saving the footprints to the design library, ensure that every Capture part is mapped to a corresponding footprint. If any Capture part is not mapped to a footprint, errors will be displayed when you run AutoECO with a design library

selected (select the Use design library only check box in the AutoECO dialog box and specify a design library).

If you save the selected footprints to an existing design library, Layout prompts you if:

- a footprint with the same name already exists in the design library. You can replace the existing footprint with the new footprint.
- a footprint is already mapped to a Capture part. You can update the mapping information.

Related topics

Creating design libraries

Connector Stagger X tab (Pad Array Generator)

Use this tab to create pad arrays that are numbered from left to right, top to bottom. This pad array style also gives you the power to add or remove a pad from alternating rows with the Row Delta setting.

X Direction area

Number (p) Number of pad columns in the X direction.

Spacing (x) Spacing between column centers in the X direction, entered in the selected units. For example, if you had a 60 mil pad and wanted a 20 mil space between the pad columns, you would enter 80 in this field. Total spacing cannot exceed 32 inches (812.8000 mm).

Start Value — The value for the first pad in the array. The start value must be entered in the entry box as a number, and is converted to numeric or alphabetic pad labels. Zero corresponds to the letter “A” when using alphabetic pad labels.

Increment — Increment at which pads are labeled. For example, if you wanted to skip from 1 to 3 to 5 and so on, you would enter a value of 2.

Numeric or Alphabetic — Sets alphabetic or numeric labeling style for the pad array. When using Alphabetic labeling, you can specify what letters to use in the Array Alphabet dialog.

Y Direction area

Number (q) — Number of pad rows in the Y direction.

Spacing (y) — Spacing between row centers in the Y direction, entered in the selected units. Total spacing cannot exceed 32 inches (812.8000 mm).

Start Value — Disabled.

Increment — Disabled.

Numeric or Alphabetic — Disabled.

Options area

Stagger (w) — Distance that every other pad in the first column is from the column line. The column line runs through the center of all pads in the column. The allowed range is –10.0000 in. (-254.0000 mm) to 10.0000 in. (254.0000 mm).

RowDelta (d)

This value specifies the addition or subtraction of a pad from the staggered row. A value of 1 will add a pad, and –1 will remove a pad.

Display Pad Name

Displays pad names in the Array Preview window.

Note: Disabling this feature will speed up display refresh rates.

Display Drill

Displays relative drill information in each pad in the Array Preview window.

Silk Screen

Allows you to set the silk screen spacing from the pin center.

The Notch check box is also provided in the Silk Screen group box. If you select this check box, the spacing will always be set wrt pin 1.

Place Outline

Allows you to set the spacing of the outline from the pin center.

Padstacks area

Default Padstack

This sets the default padstack for all pads. Use the Select button to open the Select Padstacks dialog box, and select a padstack from the libraries.

Note: Changing the Default Padstack will always set Pin 1 to the Default padstack. To use a different Pin 1 padstack, set Pin 1 after you have set the default padstack.

Pin 1 Padstack Use this to override the Default Padstack for Pin 1. Use the Select button to open the Select Padstacks dialog box, and select a padstack from the libraries.

Note: You must set the Pin 1 padstack ***after*** you have set the default padstack, otherwise your Pin 1 settings will be reset to the default.

Related topics

Pad Array Generator dialog box

Connector Stagger Y tab (Pad Array Generator)

Use this tab to create pad arrays that are numbered from top to bottom, left to right.

X Direction area

Number (p) Number of pad columns in the X direction.

Spacing (x) Spacing between column centers in the X direction, entered in the selected units. For example, if you had a 60 mil pad and wanted a 20 mil space between the pad columns, you would enter 80 in this field. Total spacing cannot exceed 32 inches (812.8000 mm).

Start Value Disabled.

Increment Disabled.

Numeric or Alphabetic Disabled.

Y Direction area

Number (q) Number of pad rows in the Y direction.

Spacing (y) Spacing between row centers in the Y direction, entered in the selected units. Total spacing cannot exceed 32 inches (812.8000 mm).

Start Value The value for the first pad in the array. The start value must be entered in the entry box as a number, and is converted to numeric or alphabetic pad labels. Zero corresponds to the letter “A” when using alphabetic pad labels.

Increment Increment at which pads are labeled. For example, if you wanted to skip from 1 to 3 to 5 and so on, you would enter a value of 2.

Numeric or Alphabetic Sets alphabetic or numeric labeling style for the pad array. When using Alphabetic labeling, you can specify what letters to use in the Array Alphabet dialog.

Options area

Stagger (w) Distance that every other pad in the first column is from the column line. The column line runs through the center of all pads in the column. The allowed range is –10.0000 in. (-254.0000 mm) to 10.0000 in. (254.0000 mm).

Display Pad Name Displays pad names in the Array Preview window.

Note: Disabling this feature will speed up display refresh rates.

Display Drill Displays relative drill information in each pad in the Array Preview window.

Silk Screen Allows you to set the silk screen spacing from the pin center.

The Notch check box is also provided in the Silk Screen group box. If you select this check box, the spacing will always be set wrt pin 1.

Place Outline Allows you to set the spacing of the outline from the pin center.

Padstacks area

Default Padstack This sets the default padstack for all pads. Use the Select button to open the Select Padstacks dialog box, and select a padstack from the libraries.

Note: Changing the Default Padstack will always set Pin 1 to the Default padstack. To use a different Pin 1 padstack, set Pin 1 after you have set the default padstack.

Pin 1 Padstack Use this to override the Default Padstack for Pin 1. Use the Select button to open the Select Padstacks dialog box, and select a padstack from the libraries.

Note: You must set the Pin 1 padstack ***after*** you have set the default padstack, otherwise your Pin 1 settings will be reset to the default.

Related topics

Pad Array Generator dialog box

Copy Padstack Layer dialog box

Use the Copy Padstack Layer dialog box to copy a padstack description from one layer to another or to add padstack

clearance to a particular layer. You might also use this dialog box to establish the padstack size on the solder layer so that you can quickly set up the solder mask.

To use the Copy Padstack Layer dialog box, you must first choose the Padstacks command from the spreadsheet toolbar button menu. With a padstack or padstack layer selected, choose the Copy Layer command after choosing Padstack from the Tool menu. Alternatively, you may choose the Copy Layer command from the pop-up menu.

Source Layer

Use this drop-down list to select the layer whose padstack you want to copy.

Target Layer

Use this drop-down list to select the layer whose padstack you want to conform to the source layer's padstack.

Oversize

Enter the value by which you want to change the padstack size. If you enter a positive integer, Layout adds this number of units to the source layer's padstack size. You may use a negative integer to subtract units from the size of the source layer's padstack.

Create Catalog dialog box

File name

- **Input** - enter the path and file name of the library or MAX file from which you want to create a catalog.
- **Output** - enter the name of the final catalog. This field defaults to the input file name plus "_llb" added as an extension.

Component Name Location

- **Above Component** - this option places the footprint name above the component in the catalog.
- **Below Component** - this option places the footprint name below the component in the catalog

Page Label

- **Label Text** - enter text that you want to display on each catalog page. This text is placed at the top of each page with the page number.
- Label Location - choose from one of three location options. Labels can only be placed at the top of each page.

Paper Size

- Choose one of eight paper sizes from the drop-down list. Due to size limitations in the Layout database, some paper sizes are not true to size. Sizes D and A1 are defined slightly smaller than the actual size of the sheet of paper.

Orientation

- Choose one of two options for paper orientation. Portrait creates the catalog page with the longest side vertically, and landscape creates the page horizontally.

Options

- **Overwrite Existing Files** - choose this option to automatically overwrite any files that have the same name in the target directory.
- **Open LOG File in Notepad** - choose this option to automatically open the LOG file after the catalog file is created. The LOG file lists all of the components

placed in the catalog files, any errors encountered, and other helpful information.

- **Open MAX File in Layout** - choose this option to automatically open the finished catalog file in Layout. Any files already open in Layout are closed after Layout prompts you to save any changes.

Create Catalog Additional Options dialog box

The Create Catalog Additional Options dialog box contains spacing, margin and sorting options for the catalog file.

Spacing Between Components

These settings designate the horizontal (X) and minimum vertical (Y) space between components on a page. Vertical spacing can vary because of differing part shapes and sizes in each row.

Spacing Between Pages

These settings designate the horizontal (X) and vertical (Y) spacing between catalog pages in the MAX file.

Sort Components By

These settings affect the order in which components are displayed in the catalog file.

- Database Order - components are not sorted, but are placed in the catalog as each occurs in the library file.
- Name - components are sorted in alphanumeric order by component name.
- Width then Height - all components are first sorted by width from smallest to largest. Then, all components of the same width are sub-sorted by height.

- Pin count - all components are sorted by pin count, beginning with the smallest number of pins and ending with the largest.

Create New Footprint dialog box

Use the Create New Footprint dialog box to name and specify the units for a new footprint. Display the dialog box by choosing the Create New Footprint button in Layout's library manager.

Name of Footprint

Enter a name for the new footprint. The footprint name can include a maximum of 255 alphanumeric characters and should not include embedded spaces.

Units

Choose English or metric units.

Use Pad Array Generator

Check this option to use the Pad Array Generator to help you create and accurately place an array of pads.

Create Stackup dialog box

Use this dialog box to specify the technology file, on which you want to base the new stackup to be created. You can display the dialog box by choosing the Define Stackup command from the Tools drop-down menu in Layout LSession window.

Design Reuse dialog box

The Design Reuse dialog is the interface for design reuse in Layout and is available from the Auto menu. The two main

areas of the dialog are the two Tree view lists. These lists allow you to specify the source and target objects for design reuse.

Select source schematic control

This list displays all of the schematics that are in the Capture design for this MAX file. The hierarchal path from the root of the design is listed for each schematic. Under each hierarchical path entry are all of the components in that hierarchical tree. Layout reads this information from the Capture netlist.

Note: Due to additions in the netlist format, you must use Capture 10.0 or newer to create the netlist. Older versions of Capture do not support Layout design reuse.

From this tree, select the schematic to use as the source for design reuse. After selecting a source schematic, the available matching target schematics appear in the Select target schematic(s) list.

Browse for source MAX file button control

This control allows you to select an external Layout MAX file as a source for external design reuse.

The external MAX file must have been created and annotated for Layout design reuse. The source schematic must be common between the two designs; otherwise design reuse will not be possible using that source file.

Repositioning the source control

This option allows you to reposition the source or initial set of components before applying the source to the target set of components.

This feature is only available with internal design reuse.

Select target schematic(s) control

This control displays all of the targets that match the selected source schematic. Check all of the targets onto which you want to apply the design reuse source. One or more targets can be selected.

OK button

Click OK to start the design reuse process. All of the components for the first target are gathered, placed and routed, then attached to the cursor for manual placement. This repeats until all selected targets have been placed. Press the ESC key at anytime to cancel the design reuse process.

Drill Chart Properties dialog box

The Drill Chart Properties dialog box appears when you choose the Drill Chart Properties command after choosing Drill Chart menu from the Tool menu. This command is available only while the Drill Drawing layer is active. To activate the Drill Drawing layer, select it from the layer drop-down list underneath the toolbar.

The values you indicate in the text boxes of this dialog box are denominated in the unit of measure you select in the System Settings dialog box. The default setting is mils. Settings in the Drill Chart Properties dialog box do not affect the size of drill chart symbols.

Drill chart text height

Use this text box to set the height of text appearing in the drill chart.

Drill chart line width

Using this text box, you can specify the width of the lines that divide the drill chart into data cells.

Related topics

Move Drill Chart command (Tool menu)

Drill Properties dialog box

The Drill Properties dialog box appears when you double-click on a cell in the Drills spreadsheet. To display the Drills spreadsheet, choose Drills from the spreadsheets toolbar button menu.

Use the Drill Properties dialog box to select the symbols you want for each drill on your drill chart. You can also use this dialog box to include comments or verbal instructions in the drill chart.

If you do not see a drill chart in the design window and you have a pre-existing design open, then the drill drawing layer is probably invisible. In order to view this layer, you need to add a color rule using the Add Color Rule dialog box.

Symbol number or letter

Use this text box to identify the drill chart symbol you want to use for a specific drill size. Appropriate values for this text box are limited to the 26 letters of the English alphabet, and the integers ranging from 1 to 20. Each integer has a corresponding drill chart symbol: the first 10 symbols (integers 1-10) are appropriate for standard through-hole drill sizes of 35 mils and greater. The second 10 symbols (integers 11-20) are the same in shape and appearance as the first 10. However, this latter group is smaller in size, and is intended for use with smaller, tightly packed drills. English alphabetic characters you enter in this text box produce a drill chart symbol formed by the corresponding letter and a small cross. Please refer to the note below for a key of text box values and their respective symbols.

The following table shows the different drill symbols available.

Drill symbol	Drill number or letter

1 and 11

2 and 12

3 and 13

4 and 14

5 and 15

6 and 16

7 and 17

8 and 18

9 and 19

10 and 20

A--Z (this example shows the A character symbol)

Drill Tolerance

Use this text box to specify a drill tolerance.

Drill Note

Notes and information you enter in this text box appear in the drill chart.

Related topics

Move Drill Chart command (Tool menu)

Drill Chart Properties dialog box

Drills spreadsheet

Dual/Quad Inline tab (Pad Array Generator)

Use this tab to create Dual or Quad Inline pad arrays. This pad array style is limited to two pad columns in the X direction, and is numbered top to bottom in a counter-clockwise direction.

X Direction area

Number (p) Disabled. (Limited to 2 columns).

Spacing (x) Spacing between pad column centers in the X direction, entered in the selected units. For example, if you had a 60 mil pad and wanted a 20 mil space between the pad columns, you would enter 80 in this field. Total spacing cannot exceed 32 inches (812.8000 mm).

Start Value Disabled.

Increment Disabled.

Numeric or Alphabetic Disabled.

Y Direction area

Number (q) Number of pad rows in the Y direction.

Spacing (y) Spacing between row centers in the Y direction, entered in the selected units. Total spacing cannot exceed 32 inches (812.8000 mm).

Start Value The value for the first pad in the array. The start value must be entered in the entry box as a number, and is converted to numeric or alphabetic pad labels. Zero corresponds to the letter “A” when using alphabetic pad labels.

Increment

Increment at which pads are labeled. For example, if you wanted to skip from 1 to 3 to 5 and so on, you would enter a value of 2.

Numeric or Alphabetic

Sets alphabetic or numeric labeling for the pad array. When using Alphabetic labeling, you can specify what letters to use in the Array Alphabet dialog.

Options area

Stagger (w)

Distance that every other pad in the first column is from the column line. The column line runs through the center of all pads in the column. The allowed range is –10.0000 in. (-254.0000 mm) to 10.0000 in. (254.0000 mm).

Stagger (z)

Distance that every other pad in the second column is from the second column line. The second column line runs through the center of all pads in the second column. The allowed range is –10.0000 in. (-254.0000 mm) to 10.0000 in. (254.0000 mm).

Display Pad Name

Displays pad names in the Array Preview window.

Note: Disabling this feature will speed up display refresh rates.

Display Drill

Displays relative drill information for each pad in the Array Preview window.

Silk Screen

Allows you to set the right spacing from the pin center.

The Notch check box is also provided in the Silk Screen group box. If you select this check box, the spacing will always be set wrt pin 1.

Place Outline Allows you to set the spacing of the outline from the pin center

Padstacks area

Default Padstack This sets the default padstack for all pads. Use the Select button to open the Select Padstacks dialog box, and select a padstack from the libraries.

Note: Changing the Default Padstack will always set Pin 1 to the Default padstack. To use a different Pin 1 padstack, set Pin 1 after you have set the default padstack.

Pin 1 Padstack Use this to override the Default Padstack for Pin 1. Use the Select button to open the Select Padstacks dialog box, and select a padstack from the libraries.

Note: You must set the Pin 1 padstack *after* you have set the default padstack, otherwise your Pin 1 settings will be reset to the default.

Related topics

Pad Array Generator dialog box

Edit Apertures dialog box

The Edit Apertures dialog box appears when you choose either New or Properties from the pop-up menu while the Apertures spreadsheet is open. (Note that only the New command is available from the pop-up menu if there are no aperture parameters selected).

With Layout, you can either create your apertures interactively, or you can include your custom aperture information in a

technology template. Layout assumes the flash is in the center of the pad.

D-code

Specifies the Gerber D-code for the aperture. The range of values for the D-code is from 10 to 500.

Width

Specifies the horizontal diameter for apertures other than thermal relief and annular.

Height

Specifies the vertical diameter for apertures other than thermal relief and annular.

Outer

Specifies the outer dimension of thermal relief and annular apertures.

Inner

Specifies the inner dimension of thermal relief apertures.

Shape

You have a choice of eight aperture shapes:

- Draw (appears as LINE in the Shape column of the Apertures spreadsheet)
- Square
- Oblong
- Oval

- Round
- Rectangle
- Thermal Relief
- Annular

Related topics

Gerber Preferences dialog box

Post Process Settings dialog box

Print/Plot dialog box

Edit Array Alphabet dialog box (Pad Array Generator)

The Edit Array Alphabet dialog appears when you click the Array Alphabet button in the Pad Array Generator dialog box.

This dialog allows you to select the letters of the alphabet that are used to label the pads in the array. These options are only used if the Alphabetic option has been selected for either the X or Y Direction.

If you choose to label your pads alphabetically, and also choose a start value of 1, your labeling sequence will start with “B”. This is due to the fact that the labeling scheme starts with 0. So, A=0, B=1, C=2, etc.

Note: By default, the alphabet selections are set to the JEDEC standard. Layout remembers the changes you make in the alphabet selections only for the current session. The next time you start Layout and open the Pad Array Generator, the alphabet selections are set to the JEDEC standard.

Standard JEDEC Alphabet button

This button resets the alphabet selections to the default JEDEC settings (A,B,C,D,E,F,G,I,J,K,L,M,O,Q,S,T,U,W)

OK

Accepts the changes and closes the dialog.

Cancel

Cancels any changes and closes the dialog box.

Related topics

Pad Array Generator dialog box

Edit Component dialog box

The Edit Component dialog box appears when you have a component(s) selected and choose:

- Properties from the Edit menu
- the Group command and then the New command from the Tool menu
- the edit toolbar button

You can also double-click a component to display this dialog box.

Reference Designator

This text box appears in the Edit Component dialog box when you select only one component to edit. Layout assigns the reference designator you enter to the relevant component.

Package

Entering a value in this text box assigns the associated electrical package, including gate and pin swap information, to the component(s) you are editing.

Value

The value you indicate in this text box is assigned to the component(s) in question.

Footprint

Choosing this button displays the Select Footprint dialog box so that you can assign a footprint to the component in question.

Location

- **X and Y location** - X and Y coordinates of the component relative to the datum of the board (the origin).
- **Rotation** - This text box specifies the rotation of the component in degrees. A component's initial orientation (zero degrees) is determined by its footprint.

Group Number

Assigns a group number in order to associate components for placement purposes.

Note: It is best to assign group numbers at the schematic level.

Cluster ID

Assigns the reference designator of the key component in a cluster as the cluster name for easy reference.

Component flags

- **Fixed** - This option fixes a component to a particular location on a board. Fixed components can neither be moved nor modified in the design window. To adjust or modify a fixed component you must edit the pertinent cell in the Components spreadsheet.

- **Not in Netlist** - Protects your mechanical parts from being deleted during AutoECO.
- **Locked** - Temporarily locks a component to a specific location so that autoplacement algorithms do not affect it.
- **Route Enabled** - Layout selects this option by default. If you clear this option, you cannot route any connections to the relevant component's pins or pads.
- **Key** - This setting designates the relevant component as a key component around which other associated components are placed.
- **Do Not Rename** - When you select this option, you exclude the pertinent component from being renamed during the automatic renaming process (initiated with the Rename Components command command from the Auto menu.).

Related topics

Select Tool command (Component)

Edit Footprint dialog box

The Edit Footprint dialog box appears when you choose the Footprints command from the spreadsheet toolbar button menu and then double-click on a Footprints spreadsheet cell.

Footprint Name

Specifies a unique footprint name used by the components to define their shapes.

Number of pads

Specifies how many pads are selected for editing.

Insert X, Y

Specifies the pad's location on a footprint. The pad location is measured from the local footprint datum rather than from the board datum.

Padstack Name

When you are editing a footprint, you indirectly determine the sizes and shapes of the pads (the padstack) by assigning a particular padstack to the pad. Any padstack listed in the Padstack Name drop-down list can be assigned to a pad. The padstack definition is given in the Padstacks spreadsheet.

Pad Entry/Exit Rule

Standard

This rule specifies that the router must route out of the pad in the direction of the longer axis of oblong or rectangular pads unless they are attached to components of three pins or less, or unless they are at the ends of DIP packages. In these cases, the router is free to route out of the sides.

Any Direction

No pad entry or exit direction rules apply for this footprint.

Long End Only

For this footprint, the router must route out of the "long end" of the pads; that is to say, in the direction of the longer axis. There are no exceptions, in contrast to the Standard option (above).

Additional Rules

Allow via under pad

By default, through vias of the same net are not allowed under SMT pads. If you wish to override this limitation, you may do so by setting this flag to Yes. Vias belonging to a different net are never allowed under pads, unless they are buried vias that will not create a short.

Preferred Thermal Relief

With this option, you can ensure that the selected pad is assigned a thermal relief when a single wire connects two pads. The exception is when a preferred thermal relief pad is connected by a single wire to a via; in which case the via is assigned the thermal relief.

A preferred thermal relief is especially useful when you have a through-hole board and you need to make connections between a capacitor and an IC. If you designate the capacitor pad as preferred thermal relief, then the appropriate pad of the pad-to-pad connection is assigned the thermal relief.

Forced Thermal Relief

If you designate any pin of a footprint as a forced thermal relief pad, it means that as long as the pin is attached to the appropriate net, the pin will get a thermal relief on the plane layers that are attached to that net.

Related topics

Edit Pad dialog box

Edit Padstack dialog box

Footprints spreadsheet

Thermal Relief Settings command

Thermal Relief Settings dialog box

Edit Free Via dialog box

When you choose the Properties command (Via) after choosing Via from the Tool menu, the Edit Free Via dialog box appears. This command is only active when you select a free via. Using this dialog box, you can modify free vias and, in the process, assign them to nets, groups, and give them via types. The Edit Free Via dialog box also allows you to select and assign a footprint to a free via. Additionally, you can lock the free via to your board with the Locked option.

Free Via *X (where X is a reference designator)*

The reference designator of the free via you choose to edit appears at the top of the dialog box.

Padstack Name

Select the name of a padstack type from the drop down list. Free vias can only be assigned padstack types that are defined in the Padstacks spreadsheet.

Net Name

Free vias must be assigned to a net, regardless of their connectivity. Use this drop-down list to designate an associated net for the free via you chose to edit.

Convert to Component

Choosing this button opens the Select Footprint dialog box. After selecting a library, choose a footprint for the relevant free via and then choose OK.

Group Number

It's possible to associate a free via with a component group while working in Layout (though it is recommended that you create groups at the schematic design level). Enter the group number you want assigned to the applicable free via.

Location

The text boxes in this settings group allow you to designate the X and the Y coordinates for the repositioning of a free via. If you leave these boxes blank and then choose OK, the free via you modified moves with your pointer until you place it on your board with a click of the left mouse button.

Locked

This option locks the relevant free via in position after you place it on a board.

Related topics

Free Via Matrix Settings dialog box

Free Via Selection Criteria dialog box

Free Via Matrix command (Place)

Edit Layer dialog box

The Properties command (Edit menu) command on the Edit menu displays the Edit Layer dialog box. You can also access this dialog box by choosing the spreadsheet toolbar button, choosing the Layers command, and then double-clicking on the layer that you want to edit.

In the Edit Layer dialog box, you can specify layer properties and designate layer pairs for mirroring.

Layer Name

This is a unique name assigned to a design layer. Layers using the same library layer or template (see below) are differentiated by number. For example, INNER3 and INNER4.

Layer name (nickname) Layer type

TOP (TOP)	Top or Component layer
BOT (BOT)	Bottom or Solder layer
INNER (INNER)	All inner routing layers
PLANE (PLANE)	Power and Ground planes
SMTOP (SMT)	Soldermask top
SMBOT (SMB)	Soldermask bottom
SPTOP (SPT)	Solderpaste top
SPBOT (SPB)	Solderpaste bottom
SSTOP (SST)	Silkscreen top
SSBOT (SSB)	Silkscreen bottom
ASYTOP (AST)	Assembly top
ASYBOT (ASB)	Assembly bottom
DRLDWG (DRD)	Drill drawing
DRILL (DRL)	Drill holes and sizes
FAB_DWG (FAB)	Fabrication drawing
NOTES (NOT)	Documentation

Layer NickName

The Layer Nickname is a three-letter abbreviation for the layer name. Every Layout layer has a nickname. This nickname can be used in place of the actual name on any dialog box that calls for a layer name, such as the Edit Layer, the Edit Obstacle dialog box, or the Text Edit dialog box.

Do not attempt to edit the layer nicknames.

Layer nicknames

- **TOP**---top
- **BOT**---bottom
- **PWR**---power plane
- **GND**---ground plane
- **IN1 through IN9**---inner layer 1 through inner layer 9
- **I10**---inner 10
- **I11**---inner 11
- **I12**---inner 12
- **SST**---silkscreen top
- **SSB**---silkscreen bottom
- **AST**---assembly top
- **ASB**---assembly bottom
- **SPT**---solderpaste top
- **SPB**---solderpaste bottom
- **SMT**---soldermask top
- **SMB**---soldermask bottom
- **DRD**---drill drawing
- **DRL**---drill layer

Layer LibName

A library layer is like a template for similarly defined layers; specifically the inner layers and the plane layers. A library layer name is the name given to this template.

For example, the library has a layer called "INNER" that represents all inner layers. Suppose that your design has layers named INNER1, INNER2, INNER3, and INNER4.

Layer library names

- TOP
- BOTTOM
- PLANE
- INNER
- SMTOP
- SMBOT
- SPTOP
- SPBOT
- SSTOP
- SSBOT
- ASYTOP
- ASYBOT
- DRLDWG
- DRILL
- COMMENT LAYER
- SPARE2
- SPARE3

Layer Type

Each layer of a Layout board design is one of six layer types. A layer's type is determined by its role within the context of a printed circuit board’s design. A layer type can apply to more than one layer of a board.

Padstack are assigned to each layer in accordance with the library layer name.

Layer Types

- **Routing Layer**. Routing layers are those layers on which all tracks for the printed circuit board should be placed, since these are the only layers for which space checking occurs. The pads on the routing layers are actual connective pads, whereas those on the plane layers are simply holes.
- **Plane Layer**. Plane layers are typically used for power and ground connectivity. Connections on these layers are assigned specific shapes and sizes that are appropriate to power and ground layers. Connections to a plane layer are clearances rather than pads. Typically, this means that they are oversized for clearance to the plane. Attaching to the plane layer with thermal reliefs is done automatically during post processing; you do not need to create the thermal pads explicitly. You cannot route on a plane layer.
- **Documentation**. Documentation layers are strictly non-electrical. These layers include soldermask, solderpaste, silkscreen, assembly, and drill drawings. No space checking is done on these layers, and no tracks should be placed on these layers.
- **Jumper Layer**. If you are routing a single-sided board using zero-ohm resistors to "jump" a track over an existing track in crowded areas, you need to define the non-routing layer as a Jumper layer. To do so, choose the spreadsheet toolbar button, choose the Layers command and double-click on the layer that you wish to use for jumpers. In the dialog box that displays, select the Jumper Layer option in the Layer Type group box. To route a single-sided board, you designate the routing side of the board as a Routing layer, and the non-routing side as a Jumper layer. The router automatically creates a jumper/via configuration on the non-routing layer when it is necessary for the completion of a connection. The via cost is set by default to 70 for two- and four-layer through-hole boards, and to 40 for almost all other multiple layer and surface mount boards. It is recommended that you set the Via Cost to approximately 90 if you want to avoid using jumpers, and to about 40 if you want to use jumpers at any convenient location. Layout automatically creates jumpers on the layer

designated as a Jumper layer. Wherever necessary, Layout creates a jumper/pin configuration in order to bridge a track on the routing layer. The autorouter places no tracks on a jumper layer; it can only place linking tracks between vias.

- **Drill Layer**. You can use drill layers to generate a drill tape, as well as to view drill holes in the design window. A Layout design may have more than one drill layer, especially if that design uses blind or buried vias, which require that drills pass between layer pairs. A layer of type Drill Layer is not the same as the Drill Drawing Layer, whose layer type is Documentation. The Drill Drawing Layer uses the hole size to access a set of drill symbols. The drill chart is generated from the Drill Drawing Layer, and you use the Drill Drawing Layer to generate a drill drawing.
- **Unused Routing**. Unused routing layers are layers being held in reserve for future use. They cannot contain any tracks. The advantage of designating an unused layer as Unused Routing rather than as Routing is that an unused routing layer uses much less of your system's memory.

Mirror Layer

The Layer Name text box indicates the layer a component, obstacle, or text item will go to if you choose the Obstacle, Opposite command from the Tool menu. You indicate which layer is to be mirrored by entering the name exactly as it appears in the left-hand column of the Layers spreadsheet. Alternately, you can enter the number of the layer as listed in the Layer Accelerator column.

Layer mirroring is especially useful when you are placing components on a surface-mount technology (SMT) board. For example, if you have a four-layer board and you want the parts on the top layer to be mirrored on the bottom layer, you can double-click on the top layer in the Layers spreadsheet and then use the Mirror Layer option to assign the solder layer as the mirror layer.

Jumper Attributes

Displays the Jumper Lengths dialog box to specify the jumper lengths and the jumper orientations that are allowed on the relevant layer.

Related topics

Drill Chart Properties dialog box

Edit Net dialog box

Thermal Relief Settings command

Design Rule Check command

Board command (Delete Violating Tracks)

DRC/Route Box command (Delete Violating Tracks)

Cleanup Design command

Edit Layer Strategy dialog box

To display the Edit Layer Strategy dialog box, choose the spreadsheet toolbar button, then choose the Strategy command and finally, choose the Route Layer command. Double-click on the sweep or the cell that you want to edit.

Sweep

If one sweep is selected, the name of the sweep for the given layer appears at the top of the dialog box. Otherwise, the number of highlighted sweeps appears.

Routing Enabled

If you select the Routing Enabled option, Layout allows you to route on a layer or layers.

Enable all probable routing layers from the outset of the design process. If you begin routing with a small number of layers, you may lack the area needed to successfully route all connections. At that point, you must enable more layers for routing. This is a reversal of the optimal design flow: it is best to start with a surplus of routing area and then decrease the number of layers in your design as routing progresses. By narrowing the number or routed layers later in the design process, you increase the chance of routing all connections successfully.

Layer Cost

The Layer Cost scroll bar determines which layers have overall preference for routing. By default, Layout assigns all layers a cost of 50. A low cost for a layer indicates that the layer is preferred for routing. A higher cost indicates that the router will avoid using that layer unless there are no other alternatives available to finish the routing of a given connection.

For through-hole boards, it is recommended that you give lower values to the outside layers (making them easier for the router to route on) as compared to the inside layers. This encourages the router to route on outside layers as much as possible. For through-hole boards it is important to have a different cost for each pair of layers. This improves the routing of busses.

For SMT (surface-mount technology) boards, it is recommended that the outside layers have a higher cost (harder for the router to route on) than the inside layers, so that the router uses a proper number of vias, rather than try to route everything on the outside of the board. The important thing for SMT boards is to be sure that all non-surface layers have the same cost, so that the router will not jam the cheaper inner layers, not leaving enough room for vias.

If a layer is disabled and the Layer Cost for that layer is set to zero, the layer is considered "restricted" (not a routing layer), and is not counted for via costing.

Primary Direction

The default strategies use only two sets of costs for Primary Direction:

- 20 (mostly vertical) and 80 (mostly horizontal) for Win/Comp, Preliminary Route, and Maze Route sweeps.
- 49 (slightly vertical) and 51 (slightly horizontal) for Next and Special Options sweeps.

Because of the increase in routing intelligence due to the addition of Between Pins cost, the old method of setting each pair of layers to a different primary direction has been superseded by the method now in place, which is to leave all layers at either 20 or 80 Primary Direction, and adjust the Layer Cost instead.

There is a special case where it is beneficial to use a strict Primary Direction cost of 10 (strictly vertical) or 90 (strictly horizontal). If you are routing a board that requires an uneven number of horizontal or vertical layers, you should set any layers that do not have an opposite direction "partner" to either 10 or 90.

Between Pins

The Between Pins scroll bar determines the cost of routing between pins on 0.100 (or less) centers. This costing works primarily on layers for which the primary direction of routing is opposite the orientation of the ICs.

The primary duty of Between Pins cost is to redirect long routes to the channels between ICs, rather then through the pins of the ICs. This discourages the router from using the channels between the long rows of PGAs and connectors.

The default Between Pins cost of 30 for the Win/Comp and Maze Route sweeps (set in the Edit Route Pass dialog box)

assigns the cost of one-half via to each "bottleneck" between IC pins.

Between Pins cost is set to zero for "Next" because any routes that are left over for the Next sweep will have a new set of paths to look at, and the possibility of finding unused channels between the IC pins.

Unless there are special manufacturing considerations that absolutely prohibit routing between IC pins, you should not set the Between Pins cost to 100. Bear in mind that a setting of 100 means you may not be able to route out of a PGA or connector.

Example #1 (AT style board)

Layer 1 (no partner) -- 90 Horizontal; Layer 2 -- 20 Vertical; Layer 3 --80 Horizontal; Layer 4 (no partner) -- 90 Horizontal.

Example #2 (uneven number of routing layers)

Layer 1 -- 20 Vertical; Layer 2 -- 80 Horizontal; Layer 3 (no partner) -- 10 Vertical; Layer 4 -- 80 Horizontal; Layer 5 -- 20 Vertical.

Related topics

Load Strategy command (File menu)

Board command (Autoroute)

Edit Route Pass dialog box

Sweep Edit dialog box

Edit Net dialog box

To display the Edit Net dialog box, select the Nets command from the spreadsheet toolbar button and double-click on the cell, row, or column you want to edit.

Net Name

The name of the selected net. If you have selected multiple nets, this text box does not appear.

Net Attributes

- **Routing Enabled** - You may manually route or autoroute the selected net only if the Routing Enabled option is enabled. It is advised that you disable routing for the power and ground nets during component placement. If you fully preroute power and ground nets, then you do not need to disable these nets while you route the remaining nets of the board. However, if you route the power and ground nets only to fanout vias, you need to disable routing of these nets.
- **Retry Enabled** - Retry Enabled allows the router to reroute the selected track to make room for another track. Although you would not normally disable Shove Enabled by itself, you might disable Retry Enabled by itself in situations where you need to keep a certain track segment on a given layer, but don't care if the router shoves the track as it routes (for example, a clock line that must be on layer 3, but does not have any critical length requirements). If Retry Enabled and Shove Enabled are both disabled, then all tracks of the net are essentially locked in place. If the net is completely routed, turning off both options is identical to using Lock. Using the Lock command (Track) affects only previously routed segments.
- **Share Enabled** - Share Enabled means that existing track within a net is considered a legal connection point for any new tracks within the net. In other words, "T" connections are allowed. Using Reconnect Enabled in conjunction with Share Enabled increases the probability of a successful routing operation. When this option is disabled, the router is forced to connect only to pads. No connections can be made to an existing track segment. You might use Share disabled to force "daisy-chain" routing for ECL boards. You would normally use Reconnect None in conjunction with Share disabled, assuming that you input the correct point-to-point netlist

into Layout (from the source through the loads to the termination).

- **Shove Enabled** - If Shove Enabled is turned on, the router is allowed to push tracks in order to make room for other tracks. You would not normally disable only shove for an existing track segment, as the router could still use Retry Enabled to rip-up the track if necessary. Therefore, if you want to completely lock a net, you should turn off both Shove Enabled and Retry Enabled. If Retry Enabled and Shove Enabled are both turned off, then all tracks of the net are essentially locked in place. If the net is completely routed, turning off both options is identical to using Lock. Using the Lock command (Track) affects only previously routed segments.
- **Highlight** - Nets that are designated as Highlight (whether routed or unrouted) are shown in the highlight color (see Color dialog box). The default color for highlight nets on all layers is white. You have the option of adding unique colors for highlighting tracks on a per-layer basis.
- **Test point** - If this option is enabled, the selected nets can be assigned a test point manually or they are assigned test points when you choose the Test Points command (Place) on the Auto menu. With the Use for test point option of the Edit Padstack dialog box, you define a via as a test point.

Group

If, in your schematic capture application, you have assigned a number to a group of nets, that number is displayed in the dialog box. The ratsnests of grouped nets is displayed in a distinct color.

All nets not assigned to a group in the schematic capture application are assigned to group zero, whose color is dark yellow. You can edit a net's group number only in the schematic capture application,

Net group numbers are associated with colors as follows:

Group	Color
Group 1	Red
Group 2	Green
Group 3	Blue
Group 4	Yellow
Group 5	Purple
Group 6	Sky Blue
Group 7	White
Group 8	Gray
Group 9	Dark. Red
Group 10	Dark. Green

Weight

Weight is the routing priority assigned to a net. The range is zero to one hundred, with 50 as the default. A higher number gives the net a higher routing priority.

A higher weight overrides all other picking order criteria. For example, the picking rule within the router states that all 50 mil tracks will be routed before any 12 mil tracks. If you set the weight of the clock lines (12 mil tracks) so that it is greater than the weight of VCC and ground, the clock lines will be routed first.

The weight of the nets is not "scaled." The only criterion that matters is whether the weight is greater or less. For example, a difference of 1 is as significant as a difference of 10.

Min Width

Min Width and Max Width define the permissible range of track widths within a net. These values are defined during

extraction by the range of widths pre-assigned to the incoming nets.

If you want to change the width of existing routes on the board, you can do so by changing the Min Width or Max Width values. For example, if you had routed a net using 8 mil tracks, and wanted to change the entire net to 10 mils, you would change the Min Width for those nets from "8" to "10."

The Force Min/Max Widths command (available on the Edit menu when the Nets spreadsheet is active) forces all of the pre-routes on a given net to conform to the range specified by the Min Width and Max Width values. Any tracks wider than the Max Width are forced down to the Max Width value; any tracks narrower than the Min Width are forced up to the Min Width value.

You can manually change to narrow tracks (and override the current Min Width) using the Route Settings dialog box and Track Width dialog box.

Conn Width

The router will create new routes using the value set by Connection Width. For nets with variable widths, set Conn Width to the preferred width. Then, you can override the preferred width as desired using the Route Settings dialog box and the Track Width dialog box.

Max Width

Min Width and Max Width define the permissible range of track widths within a net. These values are defined during extraction by the range of widths pre-assigned to the incoming nets.

If you want to change the width of existing routes on the board, you can do so by changing the Min Width or Max Width values. For example, if you had routed a net using a 50 mils track and wanted to change the entire net to 25 mils, you would change the Max Width for those nets from "50" to "25."

The Force Min/Max Widths command (available on the Edit menu when the Nets spreadsheet is active) forces all of the pre-routes on a given net to conform to the range specified by the Min Width and Max Width values. Any tracks wider than the Max Width are forced down to the Max Width value; any tracks narrower than the Min Width are forced up to the Min Width value.

You can manually change to narrow tracks (and override the current Min Width) using the Route Settings dialog box and the Track Width dialog box.

Net Layers

Displays the Layers Enabled for Routing dialog box.

Width By Layer

Displays the Net Widths by Layer dialog box.

Net Reconn

Displays the Reconnection Type dialog box.

Net Spacing

Displays the Net Spacing by Layer dialog box.

Related topics

AutoECO

Color dialog box

Colors command

Layers Enabled for Routing dialog box

Net Widths by Layer dialog box

Reconnection Type dialog box

Net Spacing by Layer dialog box

Edit Obstacle dialog box

Use the Edit Obstacle dialog box to define obstacle.

Obstacle Name

Specifies the obstacle name. If you do not specify a name for the obstacle, Layout automatically assigns a numeric name for it.

Obstacle Type

Specifies the obstacle type. The selection here determines which options described below are available for editing. The obstacle types available are:

- **Free track** - An outlined area that can be electrically attributed or electrically attached to a component pin. An outline usually displays on a routing layer and acts as a routing barrier unless the track belongs to the same net. It has no affect on placement.
- **Copper area** - A copper-filled zone that can be used for noise suppression, as a heat sink, or as a routing barrier. It can be electrically attributed or electrically attached to a component pin. It has no effect on placement. It can be filled with hatched lines or it can be solid.
- **Anti-copper** - A copper-free area within a copper pour zone.
- **Board outline** - An area that defines the board edge for routing and placement. There can be only one per board, and it is on layer 0 (All layers).
- **Via keepout** - An outlined area used by the router to define areas where vias are not allowed.

- **Route-via keepout** - An area defined by the router where routes and vias are not allowed.
- **Route keepout**. An area defined by the router where routes are not allowed.
- **Detail** - A line that is not used by place or route. It is usually used for silkscreens, drill information, and assembly drawings, and often attached to footprints.
- **Comp height keepin** - An area you define to contain all components at or above a particular height.
- **Comp height keepout** - An area you define to exclude all components at or above a particular height.
- **Comp group keepin** - An area you define to contain all components of a particular group.
- **Comp group keepout** - An area you define to exclude all components of a particular group.
- **Place outline** - An outlined area used by place routines to define the physical extent of each component on the board. It must be present in order for shove component and autoplace routines to work correctly. It is usually defined as part of a footprint.
- **Insertion outline** - An outlined area used by place routines to define the extent of pick and place.
- **Copper pour** - A filled area, identical to copper except that it features automatic voiding where there are tracks or pads. Routes can pass through it. It is recalculated at redraw.

Group

Specifies the number (between 1 and 100) of the group when the obstacle type is component group keepin or component group keepout.

Height

Specifies the height of the obstacle. This is often used with keepin or keepout areas of the board. Obstacles of the specified height and greater are affected.

Width

Specifies the width of the obstacle outline and of hatch lines for a filled or solid obstacle.

Obstacle Layer

Specifies the obstacle layer.

Copper Pour Rules

- **Clearance** - Specifies the clearance by which pads and tracks are isolated from copper pour.
- **Z order** - Specifies the location of the copper pour when it is nested or overlaps with another copper pour. The higher the Z order value, the higher control the copper pour has over other copper pours at the same location.
- **Isolate all tracks** - Specifies that any track that passes through the copper pour area is isolated from the copper pour. The isolation clearance is the largest that applies to the track.
- **Seed only from designated object** - Specifies that the copper pour won't seed from a track, and a pad designated as a seed point must be used.

Note: Layout automatically maintains the correct isolation between zones, in accordance with the established spacing rules for nets on each given layer. If you have translated a PCB 386+ board to Layout containing embedded pours, you should delete the no-fill zones after loading the file into Layout.

Net Attachment

Specifies a net attachment to an electrical obstacle for space checking and design rule checking.

Hatch Pattern

Displays the Hatch Pattern dialog box to specify a hatch pattern to the fill area for copper and copper pour.

Comp Attachment (Pin Attachment)

Displays the Component Attachment dialog box to attach an obstacle to a footprint or component.

Related topics

Edit Footprint dialog box

Edit Pad dialog box

To display the Edit Pad dialog box, choose the Spreadsheet toolbar button, choose Footprints, then double-click in the leftmost column in the Footprints spreadsheet on the pad that you want to edit.

Footprint Name

Specifies a unique footprint name used by the components to define their shapes.

Number of pads

Specifies how many pads are selected for editing.

Pad Name

Each pad of a footprint has a unique name, which is used to keep track of connectivity. The connection list uses the component name from the component list, along with the pad.

Pad X, Y

Specifies the pad's location on a footprint. This can only be set when one pad is selected. The pad location is measured from the local footprint datum rather than from the board datum.

Padstack Name

When you are editing a footprint, you indirectly determine the sizes and shapes of the pads (the padstack) by assigning a particular padstack to the pad. Any padstack listed in the Padstack Name drop-down list can be assigned to a pad. The padstack definition is given in the Padstacks spreadsheet.

Pad Entry/Exit Rule

Standard

This rule specifies that the router must route out of the pad in the direction of the long axis of oblong or rectangular pads unless they are attached to components of three pins or less, or unless they are at the ends of DIP packages. In these cases, the router is free to route out of the sides.

Any Direction

No pad entry or exit direction rules apply for this footprint.

Long End Only

For this footprint, the router must route out of the "long end" of the pads.

Additional Rules

Allow via under pad

By default, through vias of the same nets are not allowed under SMT pads. If you want to override this limitation, you may do so by setting this flag to Yes. Vias belonging to a different net are never allowed under pads, unless they are buried vias that will not create a short.

Preferred Thermal Relief

This option ensures that Layout provides a thermal relief to the selected pad within a pad-to-pad connection. However, when such a pad connects to a via, the via is assigned a thermal relief, rather than the pad designated as preferred. If you select neither the via nor the pad as preferred for a thermal relief, then the via in this situation receives the thermal relief.

A preferred thermal relief is especially useful when you have a through-hole board and you need to make connections between a capacitor and an IC. If you designate the capacitor pad as preferred for a thermal relief, then Layout assigns it a the thermal relief.

Forced Thermal Relief

If you designate a pin as a forced thermal relief, a thermal relief is provided for that pin on pertinent plane layers. If you highlight the pad in a via/pad connection and select the Forced Thermal Relief option, both the via and the pad receive a thermal relief.

When you do not specify a preference, Layout always provides a thermal relief to the via in a situation involving a via and a pad.

Related topics

Edit Footprint dialog box

Edit Padstack dialog box

Footprints spreadsheet

Thermal Relief Settings command

Edit Padstack dialog box

With the Edit Padstack dialog box, you can create a copy of a padstack, edit a padstack definition, and edit a via definition.

To access the Edit Padstack dialog box, choose the Padstacks command from the spreadsheet toolbar button menu, then double-click on the row for the padstack you want to edit.

Each padstack you use has a definition that is stored in the footprint library.

Each padstack or via definition includes specific information for each layer. If your padstack definition is not the same for all layers, you can make necessary alterations in the Edit Padstack Layer dialog box.

Each OrCAD-provided technology template defines seven padstacks for use with your board. These padstacks are designed to meet the needs of specific board types. You can add padstacks to a template by copying one of those provided, then editing the copy to suit your purposes.

Padstack

In the Edit Padstack dialog box, you can assign or change a padstack name. Each padstack must have a descriptive name of up to 100 characters. Padstack names should not contain spaces.

It is recommended that you give descriptive names to padstacks. For example, you could apply the name SQ60D32 to a .060 square pad with a drill size of .032.

Number of padstack layers

Indicates the number of padstack layers selected, when two or more layers are selected.

Non-Plated

If this option on the Edit Padstack dialog box is checked, the padstack is marked as a non-plated through hole for the manufacturing process. If a padstack is non-plated, this fact is included in the notes column of the drill chart and the Drills spreadsheet.

Use For Test Point

If this option on the Edit Padstack dialog box is selected, the selected via is defined as a test point. When you generate test points, either automatically or interactively, this via is placed. You can have as many test point vias defined as you need. In Layout, you can assign a distinctive shape or any other characteristic to your test point vias.

If you have multiple vias available for use as test points, Layout chooses the one which yields the least costly board. To do this, Layout considers pad size and whether the choice will add a drill to the board. When you choose the OK button in the Test Point Settings dialog box, the selected padstack is used for the test points that Layout creates.

Note: It is recommended that this option be assigned to vias explicitly intended for use as test points. Vias used for normal routing that are later assigned as test points may complicate test point audits. If you use routing vias as test point vias, then try to remove all test points, Layout will try to remove the routing via that is doing double duty.

Large Thermal Relief

If this option on the Edit Padstack dialog box is checked, the padstack uses the large thermal relief, as defined in the

Thermal Relief Settings dialog box, rather than the small thermal relief, which is the default.

Flood Planes/Pours

When this option is selected, thermal reliefs are flooded in copper pours.

Pad Shape

- **Round** - If you assign your pad shape as Round, the router tries to exit the pad at a 45-degree angle for best manufacturing. If you prefer that the router always use a 90-degree pad exit, set the pad shape to Oval, but maintain equal width and height. (Oval pads must exit from the "ends."). Remember that round pads must have equal width and height. If you change the size of a round pad, and change only the width, the router gives you an error message, then waits for you to change either the height or the shape.
- **Square** - Use a square pad when you want the router to exit the pad at a 90-degree angle. If you assign your pad shape to Square, the router tries to exit the pad at a 45-degree angle for best manufacturing. If you prefer that the router always use a 90-degree pad exit, set the pad shape to Rectangle, but maintain equal width and height. (Rectangular pads must exit from the "ends."). Remember that square pads must have equal width and height. If you change the size of a square pad, and change only the width, the router will give you an error message, then wait for you to change either the height or the shape.
- **Oval** - When viewed on the screen, oval pads resemble oblong pads. Some designers refer to them as "canoe" pads.
- **Annular** - You can specify that a pad is annular in the Edit Padstack dialog box. Annular pads consist of a ring of electrical connectivity with a blank circle inside: a shape like a doughnut. Annular pads are not recommended for use on plane layers.

- **Oblong** - Oblong pads will appear on the screen with the same shape as an oval, but, in fact, the mapped dimensions are those of a line segment with rounded ends. Oblong pads are often used for off--board connectors.
- **Rectangle** - Use a rectangular pad when you want the router to exit the pad at a 90-degree angle.
- **Thermal Relief** - You can specify that a padstack is a thermal pad in the Edit Padstack dialog box. Thermal relief pads are used to make connections to plane layers or to copper pour zones that can serve as heat sinks. A plane layer displays as a negative image, while a layer with a copper pour displays as a positive image. You can define your thermal pads with the Thermal Relief Settings command.
- **Undefined** - Pads on the SMT layer should be Undefined on all other layers. Blind and buried are Undefined. You do not have to specify a "zero" width and height when assigning a pad as Undefined.

No Connection

A "No connection" padstack is designed for those situations where you need to block out one or more of the layers associated with a padstack, so that Layout cannot route a connection to the pad on those layers.

A No connection padstack is normally used when a footprint requires a pad with no net connection, such as mounting holes in a board connector.

A No connection padstack can be any shape and size.

Pad Width

Pad width is the pad's measurement in the X dimension as it was originally coded onto the library shape. You must specify a width equal to the height for Round or Square pads.

If you rotate a component, the pad stacks rotate also, so that the pad width is no longer measured in the X dimension.

Pad Height

Pad height is the pad's measurement in the Y dimension as it was originally created in the library. If the pad shape is round or square, you must specify a pad height equal to the pad width.

If you rotate a component, the pad stacks rotate also, so that the pad height is no longer measured in the Y dimension.

X Offset

The X Offset is the distance from the geometric center of the pad to the track connection point.

For example, if the track connection point is on the left-hand edge of an oblong pad, that pad has a positive X offset equal to half the pad width.

Y Offset

The Y Offset is the distance from the geometric center of the pad to the track connection point.

For example, if the track connection point is on the top edge of an oblong pad, that pad has a negative Y offset equal to half the pad height.

Related topics

Edit Padstack Layer dialog box

Padstacks spreadsheet

Edit Padstack Layer dialog box

With the Edit Padstack Layer dialog box, you can edit a padstack or via definition for a specific layer.

To display the Edit Padstack Layer dialog box, choose the Padstacks command from the spreadsheet toolbar button menu and then double-click on the row for the padstack layer you want to edit.

Each padstack or via definition includes specific information for each layer. If your padstack definition is not the same for all layers, you can make edits in the Edit Padstack Layer dialog box.

Each OrCAD-provided technology template defines seven padstacks for use in your board. These padstacks are designed to meet the needs of the specified board type. You can add padstacks by making a copy of one of the seven, then editing the copy to suit your purposes.

Padstack

In the Edit Padstack dialog box, you can assign or change the padstack name. Each padstack may have a descriptive name of up to 100 characters. Padstack names should not contain spaces.

It is recommended that you create names that describe the padstack. For example, the name SQ60D32 would signify a .060 square pad with a drill size of .032.

Number of padstack layers

Specifies the number of padstack layers selected, when two or more layers are selected.

Non-Plated

If this option on the Edit Padstack dialog box is checked, the padstack is marked as a non-plated through hole for the

manufacturing process. If a padstack is non-plated, this fact is included in the notes column of the drill chart and the Drills spreadsheet.

Use For Test Point

If this option on the Edit Padstack dialog box is selected, the selected via is defined as a test point. When you generate test points, either automatically or interactively, this via is placed. You can have as many test point vias defined as you need. In Layout, you can assign a distinctive shape or any other characteristic to your test point vias.

If you have multiple vias available for use as test points, Layout chooses the one which yields the least costly board. To do this, Layout considers pad size and whether the choice will add a drill to the board. When you choose the OK button on the Test Point Settings dialog box, the selected padstack is used for the test points that Layout creates.

Note: It is recommended that this option be assigned to vias explicitly intended for use as test points. Vias used for normal routing that are later assigned as test points may complicate test point audits. If you use routing vias as test point vias, then try to remove all test points, Layout will try to remove the routing via that is doing double duty.

Large Thermal Relief

If this option on the Edit Padstack dialog box is checked, the padstack uses the large thermal relief, as defined in the Thermal Relief Settings dialog box, rather than the small thermal relief, which is the default.

Flood Planes/Pours

When this option is selected, thermal reliefs are flooded in copper pours.

Pad Shape

- **Round** - If you assign your pad shape as Round, the router tries to exit the pad at a 45-degree angle for best manufacturing. If you prefer that the router always use a 90-degree pad exit, set the pad shape to Oval, but maintain equal width and height. (Oval pads must exit from the "ends."). Remember that round pads must have equal width and height. If you change the size of a round pad, and change only the width, the router gives you an error message, then waits for you to change either the height or the shape.
- **Square** - Use a square pad when you want the router to exit the pad at a 90-degree angle. If you assign your pad shape to Square, the router tries to exit the pad at a 45-degree angle for best manufacturing. If you prefer that the router always use a 90-degree pad exit, set the pad shape to Rectangle, but maintain equal width and height. (Rectangular pads must exit from the "ends."). Remember that square pads must have equal width and height. If you change the size of a square pad, and change only the width, the router will give you an error message, then wait for you to change either the height or the shape.
- **Oval** - When viewed on the screen, oval pads resemble oblong pads. Some designers refer to them as "canoe" pads.
- **Annular** - You can specify that a pad is annular in the Edit Padstack dialog box. Annular pads consist of a ring of electrical connectivity with a blank circle inside: a shape like a doughnut. Annular pads are not recommended for use on plane layers.
- **Oblong** - Oblong pads will appear on the screen with the same shape as an oval, but, in fact, the mapped dimensions are those of a line segment with rounded ends. Oblong pads are often used for off--board connectors.
- **Rectangle** - Use a rectangular pad when you want the router to exit the pad at a 90-degree angle.

- **Thermal Relief** - You can specify that a padstack is a thermal pad in the Edit Padstack dialog box. Thermal relief pads are used to make connections to plane layers or to copper pour zones that can serve as heat sinks. A plane layer displays as a negative image, while a layer with a copper pour displays as a positive image. You can define your thermal pads with the Thermal Relief Settings command.
- **Undefined** - Pads on the SMT layer should be Undefined on all other layers. Blind and buried are Undefined. You do not have to specify a "zero" width and height when assigning a pad as Undefined.

No Connection

A "No connection" padstack is designed for those situations where you need to block out one or more of the layers associated with a padstack, so that Layout cannot route a connection to the pad on those layers.

A No connection padstack is normally used when a footprint requires a pad with no net connection, such as mounting holes in a board connector.

A No connection padstack can be any shape and size.

Pad Width

Pad width is the pad's measurement in the X dimension as it was originally coded onto the library shape. You must specify a width equal to the height for Round or Square pads.

If you rotate a component, the pad stacks rotate also, so that the pad width is no longer measured in the X dimension.

Pad Height

Pad height is the pad's measurement in the Y dimension as it was originally created in the library. If the pad shape is round or square, you must specify a pad height equal to the pad width.

If you rotate a component, the pad stacks rotate also, so that the pad height is no longer measured in the Y dimension.

X Offset

The X Offset is the distance from the geometric center of the pad to the track connection point.

For example, if the track connection point is on the left-hand edge of an oblong pad, that pad has a positive X offset equal to half the pad width.

Y Offset

The Y Offset is the distance from the geometric center of the pad to the track connection point.

For example, if the track connection point is on the top edge of an oblong pad, that pad has a negative Y offset equal to half the pad height.

Related topics

Edit Padstack dialog box

Edit Padstack Layer dialog box

Padstacks spreadsheet

Edit Place Pass dialog box

A pass is defined as a complete cycle through the current working area using one set of algorithms. In the autoplacement routine, a pass will complete one part of the batch cycle; for example, Assign Clusters, Place Clusters, or Proximity Place.

To display the Edit Place Pass dialog box, select the Spreadsheet tool and choose the Strategy command, then the Place Pass command. In the Place Pass spreadsheet, double-click on the pass or the cell you want to edit.

Use this dialog box to set the characteristics, or algorithms, Layout uses for component placement.

Pass

If one cell or row is selected, this specifies the selected pass. If two or more rows are selected, this specifies the number of selected passes.

Enabled

If you decide that the selected pass is not necessary, you can remove the check from this option to allow the batch placement routine to skip over the pass.

Done

A pass is automatically set to Done upon completion of each pass. You can use the Done flag to temporarily disable a pass, while leaving the underlying Enabled flag intact.

This offers the advantage of requiring that you need only set Done to disabled, and everything will be returned to its previous state.

This feature also allows the placement routine to "remember" where it was in the event that batch placement is interrupted. Once restarted, the placement routine will skip over any passes that are marked as "Done"

Operations

- **Assign Clusters** - If you enable this option, Layout automatically groups components together before they are actually placed on the board. This allows Layout to place the board more intelligently than would be possible

if the placement was completely random. The autoplacement tool determines component groupings primarily by optimizing the connectivity within each cluster, and minimizing the connectivity between clusters. In other words, the basic idea is to maximize the number of connections between components within a cluster, while minimizing the number of connections between different clusters.

- **Proximity Place** - The Proximity Place algorithm looks at the locations of the clusters as a starting point for the placement of the board, and then runs through thousands of permutations for each cluster, step by step settling on a placement of the board that optimizes the placement quality. Proximity Place, as its name suggests, does not produce an absolute finished position for each component. Much like a human designer, Proximity Place comes up with the approximate optimum position of each component (figuring that a final pass can adjust the placement more efficiently), rather than trying to come up with a "perfect" placement before every component is in its place.
- **Adjust Components** - Spreads components that are bunched together, and aligns things aesthetically after the bulk of the placement work is done.
- **Place Clusters** - This places the assigned clusters in the best location on the board, relative to the other clusters and any fixed components. For example, a cluster containing buffers would generally end up near the I/O connectors, memory would end up clustered together in another area, and heavily interconnected logic would end up in another area.
- **Swap Comps** - If you choose this option on the Place Pass dialog box, Layout swaps adjacent components in order to refine a nearly completed placement.
- **Swap Pins** - Swap pins uses the information in Packages from the original schematic to allow automatic pin swapping during autoplacement. This is not allowed if you did not come into Layout with a schematic.

Options

- **Fast Reconnect** - For greater speed, the standard strategy uses the Fast Reconnect flag for Proximity Place. If you are running a long "weekend" batch, you can turn off the Fast Reconnect option to achieve a slightly better quality placement. Swap components also uses the Fast Reconnect option. The standard placement strategies are set up using these general guidelines
- **Swap Gates** - Specifies to automatically swap gates in order to reduce the overall connection length on the board.

Iterations

The Iterations scroll bar determines how many different algorithms Layout will employ during a given placement.

If you want to try many different algorithms on a given board, set the Iterations count to a high number.

If you want to attempt a few different algorithms (in order to get a quick result), set the Iterations count to a low number. The higher the Iterations, the better the chances that Layout will find the optimum algorithm for the board in question.

Attempts

The Attempts scroll bar determines how many different placements the autoplacement tool will undertake during each of the Iterations.

If you want to try many different placements on a given board, set the Attempts count to a high number. If you just want to attempt a few different placements (in order to get a quick result), set the Attempts count to a low number. The higher the Attempts, the better the chances that Layout will find the optimum placement for the board in question.

Max Clusters

The Max Clusters scroll bar determines how many clusters Layout uses while organizing the components on the PCB for placement.

The default of 10 clusters will usually result in the optimum placement, but as few as 5 clusters per 100 IC equivalents is acceptable.

Edit Route Pass dialog box

You can enable and disable sweep passes using the Route Pass spreadsheet. You can also access and modify some of the routing parameters set in your strategy file, such as via cost, retry cost, route limit, and attempts. To display the Edit Route Pass dialog box, select the Spreadsheet tool, choose the Strategy command, choose the Route Pass command, then double-click on the pass or the cell that you want to edit.

Pass

If a single pass is selected, specifies the name and number of the pass. If two or more passes are selected, specifies the number of selected passes.

Enabled

A Route Pass is defined as a complete cycle through the current working area using one set of algorithms. In Layout, this means that the router will route all of the connection that it can within a routing window. If you are in batch mode, this will be repeated throughout the board. In Layout, enabling a pass enables the corresponding sweep.

You will notice that only one pass is enabled per sweep. The other two passes are listed for convenience in using various

cost settings and routing algorithms without having to "invent" the pass from scratch.

Done

A pass is automatically set to "Done" by Layout upon completion of each pass.

You can use the Done flag to temporarily disable a pass, while leaving the underlying Enabled flag intact.

The advantage to this is that you need only set Done to enabled, and everything will be back the way it was.

Type

For each pass, you can choose one of the following routing types:

- **Heuristics** - Heuristics is one of seven routing type options available on the Edit Route Pass dialog box. A Heuristic sweep ignores all costing except Attempts. One Attempt will route only "pure memory." More than one Attempt will also route "near memory" and "straight hits." Heuristics will only route connections for which the end points are exactly horizontal or vertical to one another or for which there are one or fewer connections between endpoints.
- **Maze** - Maze is one of seven routing type options available on the Edit Route Pass dialog box. Maze routing enables all of the shove and retry capabilities of the router. All cost settings are active when you use the Maze option. One Maze sweep should route most boards to very near completion.
- **Auto DFM** - Auto DFM (Design for Manufacturability) is one of seven routing type options available on the Edit Route Pass dialog box. Selecting this option for the Route pass is identical to invoking the Cleanup Design command from the Auto menu immediately after you complete the routing of your board. When you select the Auto DFM option in the Route Pass dialog box, Layout

invokes the Auto DFM capability after the route is complete.

- **Fanout** - Fanout is one of seven routing type options available on the Edit Route Pass dialog box. It is advisable, in most cases, to use the Fanout command available on the Auto menu. The Fanout router will route most SMT pads to through-hole vias. This router uses a simple Heuristic algorithm to search for a legal location "inside" the IC first, and if that is not clear, "outside" the IC. In some cases, especially on double-sided SMT boards, neither path may be clear. In that case, the router will drop the connection and go on to the next one. If you are trying to reserve space for vias, you can ignore any failures at this point, because the Maze Route sweep will pick them up later. If you are routing VCC and ground to vias, or if all SMT pads must be dispersed, you should inspect the board after running the fanout router, and use the Route Settings dialog box to finish the last few fanouts. You can implement fanout for power and ground pins using the procedure described in Implementing power and ground fanout

- **Via Reduce** - Via Reduce is one of seven routing type options available on the Edit Route Pass dialog box. Use the VIARED_H.SF or VIARED_V.SF strategy files to run a Via Reduction sweep on your fully routed board. You do not need to run via reduce in most circumstances, as the router by its nature will minimize vias as it progresses through the routing operation.

- **Auto CDE** - Auto CDE is one of seven routing type options available on the Edit Route Pass dialog box. AutoCDE (Clear Design Error) is an "unrouter" that removes all shorted tracks so that the autorouter will have a clean design from which to reroute the board. This allows you to move components on your host system without worrying about creating shorted tracks. When you re-enter Layout after an engineering change order, you would first run an AutoCDE sweep through the board, automatically removing all shorted tracks. Then you would route the board normally, with assurance that there are no pre-existing spacing violations.

Options

- **Partial** - If Partial is enabled, the router will route a net from its source up to the edge of the routing window. A ratsnest extends from the end of the partial route to the net destination. The remainder of the net is routed when the routing window moves to that area of the board that contains the ratsnest. You are most likely to use No Partials when you are using Route Window on an analog section of the board, and you want the router to ignore any digital connections that pass through the window.
- **Fast** - Fast Route is recommended for quick routing or a placement check on a board. Fast Route is not recommended for production board routing. Use the FAST_H.SF or FAST_V.SF strategy files to run a quick "placement check" route on a board.

Via Cost

With the Via Cost value in the Edit Route Pass dialog box, you set the probability that the router will use vias as it routes the board. A higher Via Cost causes the router to use fewer vias. As a general rule, (with all other costs set to their defaults) on a two-layer board the following guidelines apply:

- A 40 Via Cost means that the router will use a via if it has to go 0.250" out of its way. Recommended for use only in routing VCC and ground nets, or for a final try at completing a board.
- A 50 Via Cost means that the router will use a via if it has to go 0.300" out of its way.
- A 70 Via Cost is the default. This means that the router will use a via if it has to go .400" out of its way.
- A 100 Via Cost prohibits vias.

The more layers in the design, the more reluctant Layout will be to use a via. If you wish Layout to ignore a disabled layer for via costing (in other words, route the remaining layers as if the disabled layer did not exist), you should both disable the layer and set the Layer Cost to zero.

Retry Cost

There are no hard and fast rules as to when Layout will use Retry and when the router will try to go around the existing track instead. However, there are some general guidelines:

- A 30 Retry Cost means that the router will try once to route around any existing track, and if unsuccessful, will retry the route on about its second attempt.
- A 50 Retry Cost means that the router will try three times to route around any existing track, and if unsuccessful, will retry the route on about its fourth attempt.
- A 60 Retry Cost means that the router will try nine times to route around any existing track, and if unsuccessful, will retry the route on about its tenth attempt.
- A 100 Retry Cost prohibits any retry of existing track (this will also save RAM).

You should use low Retry Cost with high Via Cost. This enables the router to correct "wrong ways" that block too many routes.

You should use high Retry Cost with low Via Cost. This keeps the via count from getting too high due to excessive retry of the few wrong ways that would be present with a low Via Cost.

You should use a relatively low Retry Cost at the beginning of the design, in order to give the router freedom to reroute existing routes more efficiently in relation to the newer tracks being installed.

You should use a higher Retry Cost at the end of the design, to cut down on useless Retry attempts, when it would be easier to route around the existing track instead of through it.

Route Limit

The Route Limit value determines the amount of effort the router expends trying to route a particular track. A higher Route Limit indicates more effort given to finding a path for the track and less attention to route efficiency or quality. The

Route Limit value that OrCAD sets as a default very rarely needs to be altered.

- A 20 route limit will route only extremely "clean" and efficient routes. This is not recommended, unless you plan to do a substantial amount of interactive routing, and you just want the router to start the design.
- A 50 route limit will route 90% of most designs. This is not recommended, unless you plan to do a substantial amount of interactive routing, and you just want the router to start the design.
- An 80 route limit will autoroute most designs to completion, as quickly as possible, without excessive numbers of vias. This is the standard default value set by OrCAD.
- A 100 route limit should be used only to finish a design, as there is no limit to what kind of tracks the router will place (little attention to quality).

Attempts

With the Attempts scroll bar in the Edit Route Pass dialog box, you set the number of times that the router will attempt to route a connection. In Layout, Attempts can have an affect on both time to completion and completion percentage.

On a board that the router is capable of routing to 100% (it's best to assume this is the case any time you begin a board) too few attempts will interrupt the connection order, causing the router to fail on some connections that would otherwise have been routed, given enough attempts.

Typically, in a window of 100 connections, 90 will be routed by the second attempt. Five of the connections will take five attempts, three connections will take 10 attempts, and the remaining two might take 15 or 20 attempts.

Twelve attempts (the default) is sufficient to route almost any routing window to 100% or very near, allowing each connection to be routed in its proper order, without taking an inordinate amount of time. Although this may leave a few

connections unrouted, they will be picked up during the Next sweeps (with up to 100 attempts).

Related topics

Edit Route Pass dialog box

Fanout

Cleanup Design command

Edit Spacing dialog box

To invoke the Edit Spacing dialog box, choose the spreadsheet toolbar button, choose the Strategy command, and then choose the Route Spacing command. Double-click on the layer or the cell that you want to edit in the Route Spacing spreadsheet.

Layer

If you selected a single layer, the name of the selected layer appears at the top of the dialog box. If you selected multiple layers, the number of layers appears at the top of the dialog box.

Track to Track Spacing

Track-to-track spacing specifies the minimum space required between tracks of different nets, and between tracks and obstacle of different nets.

Note that the generic track-to-track spacing set here can be overridden on a "per-net" basis using the Net Spacing By Layer dialog box, which is accessed from the Net Dialog.

Track to Via Spacing

Track-to-via (and obstacle-to-via) spacing specifies the minimum space required between vias and tracks of different nets.

Track to Pad Spacing

Track-to-pad (and obstacle-to-pad) spacing specifies the minimum space required between pads and tracks of different nets.

Note: You need to set Track to Pad Spacing to "7," not "8" when using 8 mils tracks with an 8 1/3 grid and 60 mils pads. The actual spacing will be 7 2/3 mils.

Via to Via Spacing

Via-to-via spacing specifies the minimum space required between vias of different nets.

Via to Pad Spacing

Via-to-pad spacing can be used to specify the minimum space required between pads and vias of the same net (as well as different nets, which is the usual case).

For example, if you wish to keep a distance of 25 mils between your SMT pads and the fanout vias that are connected to the pads, set Via to Pad Spacing to 25.

Pad to Pad Spacing

Pad-to-pad spacing specifies the minimum space required between pads of different nets.

Related topics

Edit Net dialog box

Edit Test Point dialog box

The Edit Test Point dialog box appears when you choose the Properties command (Test Point) from the Tool, Test Point menu. Using this dialog box, you can modify existing test points and in the process, assign them to nets, groups, and give them via types. The Edit Test Point dialog box also allows you to select and assign a footprint to a pre-existing test point. You can also lock a test point to your board with the Locked option.

Test point *X (where X is a reference designator)*

The reference designator of the test point you're editing appears at the top of the dialog box.

Padstack Name

Select the name of a padstack type from the drop down list. Test points can only be assigned to padstacks that are both defined and enabled for use as a test point.

Net Name

Use this drop-down list to designate an associated net for a new test point.

Convert to Component

Choosing this button opens the Select Footprint dialog box. After selecting a library, select a footprint for the test point you're editing.

Group Number

It's possible to associate a test point with a component group while working in Layout. Enter the group number you want to assign to the modified test point.

Location

The text boxes in this settings group allow you to designate X and Y coordinates for the positioning of the modified test point.

Locked

This option locks the relevant test point in position after you place it on your board.

Related topics

Edit Padstack dialog box

Properties command (Padstack)

Fanout Settings dialog box

The Fanout SMD Pads dialog box displays when you choose any of the fanout commands from the Auto menu. You use this dialog box to specify how Layout implements the fanout operation.

Note: In previous versions of Layout, fanout capabilities were referred to as "via dispersion." In addition to the change in terminology, the implementation procedure has changed as described in the online help.

Power/Ground

- **Fanout Power/Gnd** - If selected, Layout implements fanout for power and ground SMD. Power and ground pads are identified by their being enabled on an appropriate plane layer in the net spreadsheet.
- **Lock after fanout** - If selected, fanout routes and vias for power and ground nets are locked after fanout is complete. This prevents the autorouter from moving the fanout vias farther away from their respective pads.

- **Disable after fanout** - If selected, power and ground nets are disabled after fanout is complete. This is especially advantageous if you plan to perform a batch route after fanout is complete.

  If Power\Ground Fanout fails to complete all pins, the nets will not be disabled, and Layout will display a notification.

- **Share close vias** - If selected, routes that belong to a single power or ground net can share a single via. Note that via sharing can result in long fanout routes or large currents.

- **Use free vias -** If selected, free vias can be used for optimal implementation of power and ground fanout.

Signals

- **Fanout Signals** - If selected, Layout implements fanout for signals connected to SMD pads. A signal connection is any net that is not enabled on a plane layer.

- **Lock after fanout** - If selected, signal routes and vias are locked after fanout is complete. In general, it is best to leave signal routes unlocked, so the autorouter can move them as necessary to complete routing the board.

- **Share close vias** - If selected, routes that belong to a single net can share a via. Via sharing for signals reduces the number of vias for (and, therefore, the congestion of) the board.

- **Use free vias** - If selected, free vias can be used for the optimal fanout of signals connected to SMD pads.

IC Fanout Direction

- **Inside** - Fanout vias are allowed inside (or under) the SMD.

- **Outside** - Fanout vias are allowed outside the SMD.

Maximum Fanout Distance

The value you set for this option determines the maximum distance from the SMD pad at which Layout will place the fanout via. This distance is the Euclidean distance (measured from the center of the SMD pad), not the cumulative distance of the route segments. Layout will only place vias on grid points, so the actual distance from pad to fanout via may be slightly longer than the specified distance, since distances are rounded up. By default, the value for this option is 300 (mils).

Default via

If you have assigned a via to a specific net, using the Via per net command (and if you don't have the Override via per net check box selected), that via will be used when fanning out. If you have not assigned a via using the Via per net command, Layout uses the via named in this list box when fanning out.

Override via per net

Use this option to override the via assigned using the Via per net command with the via named in the Default via list box.

Related topics

Fanout

Board command (Fanout)

Component command (Fanout)

DRC/Route Box command (Fanout)

Implementing power and ground fanout

Edit Layer Strategy dialog box

Board command (Autoroute)

Thermal Relief Settings command

Fanout tab (SPECCTRA Automatic Router Parameters)

Direction

Directs SPECCTRA to escape wires and vias relative to the component pins.

In	Escapes wires and vias inward from the component pins.
Out	Escapes wires and vias outward from the component pins.
Either	Escapes wires and vias inward or outward from the component pins.

Via Location

Directs SPECCTRA to escape wires and vias relative to the component outline.

Inside	Inside the physical component outline.
Outside	Outside the physical component outline.
Anywhere	Either inside or outside the component outline.

Use the Via Location options together with the Direction options to control fanout direction. Using these options together enables fanout to locate vias relative to both the component pins and the physical component outline. For example where the component outline extends beyond the component pins, and where manufacturing or test requirements demand that fanout vias be accessible, the combination of direction out and location outside would ensure that fanout vias were directed outward from pins and beyond the physical component outline.

Max Fanout Length

Restricts the routed length of the escape wires.

The maximum length is measured from a pad's origin to the center of the via. The default is -1, which means there is no restriction on the routed length.

Fanout Grid

Specifies a temporary fanout via grid, used only when the Fanout command is executed.

1 Wire Between Vias	Sets a via grid that allows one wire to be routed between adjacent vias.
2 Wire Between Vias	Sets a via grid that allows two wires to be routed between adjacent vias.
Current Via Grid	Uses the via grid defined in the Router Setup tab of the SPECCTRA Automatic Router Parameters dialog box.
Specified Grid	Enter a value that SPECCTRA uses as the uniform X, Y via grid. SPECCTRA creates equidistant grid points in the X and Y directions.

Pin Types

Specifies the types of pins that are escaped.

All	All component pins are escaped.
Specified	The pins types you select are escaped. Select the check box next to the pin types you want to escape and deselect the check box next to the pin types you do not want to escape.

All

All component pins are escaped.

Power Nets

Select this check box if you want to escape all pins that have power nets assigned.

Signal Nets

Select this check box if you want to escape all pins that have signal nets assigned and that interconnect with one or more other pins.

Single Pin Nets

Select this check box if you want to escape all single pin signal nets.

Sharing

Controls pin and via sharing.

Share Within Distance

Sets the maximum distance that a via or pin can be from a through-pin or via if the Share Pins or Share Vias check boxes are selected.

Vias and pins farther away from these pins will not share a fanout via.

The default is -1, which means pin sharing can occur with any pin or via within the default distance of 200 mils. If you enter a value of zero, a maximum distance is not set.

Share Pins

Controls whether SPECCTRA can escape to through-pins on the same net.

- If you select this check box, SPECCTRA escapes to a through-pin if the cost is lower than the cost of using a via and the pin is within the distance specified in the Share Within Distance field.
- If you do not select this check box, SPECCTRA is forced to use only vias for escapes.

Max Share Count

Controls the maximum number of connections that can attach to shared pins when the Share Pins check box is selected.

If you select this check box, you must enter a limit in the text box. If this check box is not selected, any number of connections are allowed.

Share SMD's on Way to Via

Controls whether SPECCTRA can connect SMD pins on the same net before escaping to a shared pin or via.

- If you select this check box, SPECCTRA escapes each SMD pad directly to a pin or via if the cost is lower than the cost to escape directly to a pin or via.
- If you do not select this check box, SPECCTRA is forced to escape SMD pins directly to a pin or via.

Max Share Count

Specify the maximum number of SMD pads that can be connected to a shared escape wire when the Share SMD's On Way to Via check box is selected.

If you select this check box, you must enter a limit in the text box. If this check box is not selected, any number of connections are allowed.

Share Vias

Enables SPECCTRA to share vias between SMD pads on the same net.

If this check box is deselected, SPECCTRA uses unique vias for every surface mount pad.

Max Share Count

Specify the maximum number of connections that can attach to shared vias when the Share Vias check box is selected.

If you select this check box, you must enter a limit in the text box. If this check box is not selected, any number of connections are allowed.

Find and Select Item dialog box

The Find and Select Item dialog box appears when you choose any one of the Select From Spreadsheet commands available in the Tool menu. Layout regards the value you enter in the Find text box as case-sensitive.

Find

Use this text box to specify the object or entity you want to find and select in the relevant spreadsheet. Remember that all queries are case-sensitive. Use the asterisk character (*) or

the question mark character (?) as a substitute for a string or a single character, respectively.

Related topics

Query Window command (View menu)

Component Selection Criteria dialog box

Find Coordinate or Reference Designator dialog box

The Find Coordinate or Reference Designator dialog box appears when you choose the Find/Goto command (Edit menu) from the Edit menu.

Item Name

The Item Name can be a component name and reference designator separated by a period (for example, U1.1), or a set of X and Y coordinates separated by a comma (for example, 1200, 4300).

Related topics

Query Window command (View menu)

Find and Select Item dialog box

Component Selection Criteria dialog box

Footprint Selection dialog box

This dialog box appears as a result of selecting the Alternate Footprint command (on the pop-up menu while the Component tool is active).

Select New Footprint

This option consists of the last footprint you chose to replace the currently selected component. If you never selected a footprint to replace the selected component, then the option that appears in the Footprint Selection dialog box is that of the selected component.

Related topics

Edit Component dialog box

Free Via Matrix Settings dialog box

Padstack Name

Use this drop-down list to specify a padstack type. A padstack is available on this drop-down list only if it is defined in the Padstacks spreadsheet. The default padstack is VIA1.

Net Name

You must assign free via matrices to a net. The nets available in the Net Name drop-down list are those defined in the Nets spreadsheet. The default net is GND.

Group Number

This text box allows you to assign all of the vias that you create to a particular component group. Zero (no group) is the default group.

Minimum X Pitch

This is the minimum horizontal distance between free via columns. Via columns can be farther away from each other if necessary, but they cannot be any closer together than this value.

Minimum Y Pitch

This is the minimum vertical distance between free via rows. Via rows can be farther away from each other if necessary, but they cannot be any closer together than this value.

Via to Edge Space

The value you enter in this text box indicates the amount of distance you want between the center of a via and the periphery of its associated copper area (when the Free Via Matrix command (Place) is used to place vias inside a copper area).

Spacing Tolerance

This is the allowable spacing tolerance between the center of the via and the outer edge of the copper area. The larger the Spacing Tolerance value, the easier it is for Layout to insert the via matrix (ideal placement is less likely, however).

Lock Free Vias

This option locks the matrix after placement.

Periphery Only:

This option places vias around the periphery of the free via matrix area, but does not allow the placement of any free vias in the center of the designated area.

Related topics

Free Via Selection Criteria dialog box

Edit Free Via dialog box

Free Via Selection Criteria dialog box

This dialog box appears as a result of choosing the Select from Spreadsheet command from the Tool, Via menu. The Components spreadsheet opens simultaneously.

You can limit or broaden your search for a free via with the three drop-down lists and Exclude Locked option. When you choose the OK button, the free vias that meet your criteria are located in the Components spreadsheet and highlighted.

Padstack Name

From the options presented in this drop-down list, select the padstack associated with the free vias you wish to highlight.

Net Name

Although a free via you create may not affect connectivity, you must nevertheless assign it to a net. To highlight free vias associated with a particular net, select the appropriate net name from this drop-down list.

Group Number

Layout regards free vias as components; just as with components, you have the ability to assign free vias to a group. To find and highlight free vias associated with a specific group, enter the relevant group number in this text box.

Exclude Locked

Just as with components, you may lock free vias in place on a printed circuit board. If you want to exclude all locked free vias from the selection process, enable this option.

From Layout v9.0, you can define the specifications of a free via just as you are able to define vias and padstacks.

Related topics

Free Via Matrix Settings dialog box

Edit Free Via dialog box

GenCAD to Layout dialog box

Input GenCAD File

In this text box, enter the path and filename of the GenCAD file you want to translate into a Layout board (.MAX) file.

Output Layout File

In this text box, enter the path and filename of the Layout file you want to create with a translated GenCAD file.

Overwrite existing files

By selecting this option, you dispense with any warnings that Layout normally provides before overwriting files.

Input Technology File

Enter the name of a technology file (.TCH) in this text box. The technology file serves as a design template.

Circumvent VeriBest bug

Related topics

Layout to GenCAD dialog box

General tab (SPECCTRA Automatic Router Parameters)

SPECCTRA path

Enter the path to the SPECCTRA.EXE file in the SPECCTRA installation that you want to use for routing your board, or click Browse to select the SPECCTRA.EXE file.

If you click the Browse button, the Find SPECCTRA for you? message box appears.

Click	To
Yes	Display the Find SPECCTRA dialog box. The Find SPECCTRA dialog box displays the versions of SPECCTRA.EXE found in the directories listed in the PATH environment variable. Do one of the following: ■ Select the SPECCTRA version you want to use and click Select. ■ Click Cancel to manually browse for the SPECCTRA.EXE file.
No	Manually browse for the SPECCTRA.EXE file.

Use this program to edit DO file

Enter the path and filename of the text editor you want to use for editing the customized .DO file or click Browse to select the executable for the text editor.

The default text editor is Notepad.

Use a customized DO file

Select this check box if you want to use a customized .DO file for routing your board using SPECCTRA.

Enter the path and file name of the customized .DO file, or click Browse to select the .DO file.

Create

Click Create if you want to create a .DO file with the default options that you can use to run SPECCTRA from Layout. The SPECCTRA template DO file dialog box appears.

1. Enter the DO file name in the File name field and click Save to create the customized DO file.
2. In the Would you like to edit the new DO file dialog box, click Yes to edit the new file with Notepad.

   There are many commands that are included, but not enabled in the default .DO file. To enable these commands, remove the comment (#) from the respective command line. Do not remove the following command from the DO file.

   ```
   write routes $/SpecctraWithinLayout.RTE
   ```

   This command tells SPECCTRA to write the route (RTE) file in the same directory as your board. The file must be named SpecctraWithinLayout.RTE so that Layout can automatically reload your design after you close SPECCTRA.
3. Save the changes you make and exit Notepad.

Edit

Click Edit if you want to edit the specified .DO file.

Include Advanced Per Layer/Object Spacing Rule

Select this check box if you want pcb layer spacing rules and object spacing rules (like line to line and via to via spacing rules) to be passed to SPECCTRA.

Note: The layer and object spacing rules will be passed to SPECCTRA only if have installed the SPECCTRA license with the ADV option

Generate Reports dialog box

The Generate Reports dialog box appears when you choose the Create Reports command from the Auto menu.

Select Reports to be Generated

Each report type available to you has its own option; select one or more to create a text file of each selected type. You can then either view these text files immediately with your default text editor (available through the Text Editor command (File menu) on the File menu), or you can save these reports as plain text files and place them in a local directory of your choosing (see below).

Select from the following report types:

- **Comp All (Comps) -** a list of components in alphanumeric order, one per line. Information pertaining to each component can be included to the right of the component name. For information on how to customize this report, see Component list and netlist report definition.
- **Comp Bottom SMT (Smbot)** - a list of surface-mount components on the bottom layer of the design, along with their associated part numbers, part names, and footprints.
- **Comp Bottom Thru (Thbot)** - a list of reflected, through-hole components, along with their associated part numbers, part names, and footprints.

- **Comp Insertion (Insert)** - a special, predefined version of the user-defined component list that shows component insertion locations.
- **Comp Top SMT (Smtop)** - a list of surface-mount components on the top layer of the design, along with their associated part numbers, part names, and footprints.
- **Comp Top Thru (Thtop)** - a list of unreflected, through-hole components, along with their associated part numbers, part names, and footprints.
- **Connections (Conn)** - a list of point-to-point connections in each net for testing point-to-point connectivity.
- **Conns Unrouted (Unroute)** - a list of unconnected routes in the design and the X and Y coordinates of each terminal point in the unconnected routes.
- **Cross References (Xref)** - a cross-reference list of component pins and the nets attached to the pins. This list also includes pin locations for testing purposes.
- **Drills (Drill)** - a human-readable drill list separated by drill layer pairs. To generate an Excellon drill tape, choose the Post Process toolbar button, then choose Drill Tape. The drill tape report has a .TAP extension.
- **Drill Pairs (Lev1_Lev2)** - a report of all blind and buried vias between any two specified layers. For example, 2_5.DRL reports data for all vias that exist on or between layers 2 and 5 in the design. The through-hole drills have their own file, with the name THRUHOLE.DRL. The data in this report includes the drill size, drill tool, and the X and Y coordinates for the vias, and it matches that of the drill tape.
- **Net Lengths (Netlen)** - a list of nets with net length information.
- **Net List (Netlist)** - a list of nets in alphanumeric order, one per line. Information pertaining to the net can be included to the right of the net name. For information on how to customize this report, see Component list and netlist report definition.

- **Padstacks (Padstack)** - a list of all padstacks in the design, and the footprints and components that use them.
- **Part List (Partlist)** - a list of part numbers and part types (derived from the schematic), followed by a list of all components using each combination of part number and part type.
- **Pins Unused (Unusepin)** - a list of unused pins on each component.
- **Renames (Rename)** - a list of all component renames and gate and pin swaps, used for documentation purposes and for manual back annotation for non-supported schematics. If a back annotation was performed, this list may be empty.
- **Statistics (Stats)** - a list of the information in the Statistics spreadsheet presented in a report format.
- **Test Points (Tpoint)** - a list of test points and their locations.
- **Vias (Vias)** - a list of all vias and their locations.

View Reports

If you select this option, the default text editor of your system displays the reports you selected in the Select Reports to be Generated settings group. No files are saved to your local directory and the three file options to the left of the View Reports option are disabled.

Save As Files

Append to existing reports of same kind

- Use this option to add newly created reports to existing files of the same type.

Use default file names

This option indicates that Layout is to use predefined names for the reports you create. These predefined names appear in parentheses in the Select Reports to be Generated settings group. All file names end with the .TXT extension.

Use Current Design Directory

Selecting this option places newly created reports in the current design directory (the directory containing the board file you are currently working on).

Browse

The Browse button is enabled if you clear the Use Current Design Directory option. When you choose the Browse button, a standard Windows dialog box appears, allowing you to select the directory where you want report files saved.

Save Settings

When you choose the Save Settings button, any settings that you selected are saved. The next time you choose Create Reports command, the Generate Reports dialog box appears with the settings you previously selected and saved.

Select Custom Reports

- Net Properties Report (Netprop)

Highlighting this option creates a report that lists net information, including net names and lengths.

- Component Properties Report (Compprop)

Highlighting this option creates a report that lists component information, including component reference designators and values.

Gerber Preferences dialog box

The Gerber Preferences dialog box appears when you choose Gerber Settings from the Options menu. If you choose the Save Gerber Preferences button, the settings in this dialog box are saved independently of your design (in the LAYOUT.INI file), so that they remain from one Layout session to the next.

Aperture Settings

Maximum Apertures

This designates the maximum number of apertures your plotter allows. The default setting is a value of 999. You can specify a range between 24 and 2,001.

Gerber Creation

Create Apertures as Needed

This is the default option; it creates apertures using available D-codes as they are needed to post process the design. When creating a 274D Gerber with this feature enabled, the created aperture file may contain a D-code with a smaller size than your Gerber photoplotter can use. Fractional mil apertures may even be rounded to 0.0000. Always check the aperture file to verify that all apertures are usable – usually 2 mils or larger.

Use Existing Apertures Only

This setting limits post processing to the apertures already present in your board file (.MAX). Any needed apertures are simulated using those available (for example, two draws offset by 5 mils using a 10 mils line would be used to simulate one 15 mils draw). If it is not possible to simulate a given aperture, Layout notifies you that it must create the needed aperture.

One way to use this option is to load a standard technology or template file, bring up the Apertures spreadsheet, and then either create a list of apertures manually or read one in using the Read GerbTool Apertures command on the Apertures spreadsheet's pop-up menu. If you are creating an aperture list for the first time, you can save your aperture list using the Write GerbTool Apertures command on the Apertures spreadsheet's pop-up menu so that it can be reused with another design.

Using Master Aperture List

This option is similar to the Use Existing Apertures Only option, except that it uses an external GerbTool-format aperture list (with an .APP extension) to generate apertures for your design.

Retain D-codes from Master List

This option (available only when you select the Using Master Aperture List option) prompts the use of the D-codes from the master list. For example, if D143 is a 60 mils round aperture in the master list, Layout uses D143 for a 60 mils round pad. If this option is not selected, Layout assigns D-codes in the order they are used, starting at D10.

Master List

This text box and the corresponding Select Master List button (both available only when you select the Using Master Aperture List option) allow you to specify the name of the external GerbTool-format aperture list (with an .APP extension) that you want Layout to use to generate apertures for your design. The Select Master List button lets you browse for the appropriate file.

Gerber Settings

Xsize, Ysize

The largest plot or image the Gerber device can accommodate (in mils).

End-of-Block Character

Defines the character that divides Gerber commands.

Incremental

Specifies to output data relative to last move. If not selected, Layout outputs data in absolute coordinates.

CR After Each Block

If checked, the Gerber output has a carriage return after each block.

Output Resolution

Either 2.3 format (mils) or 3.4 format (10th mils). Not all photoplotters support 3.4 output. The 3.4 format is required to generate true arcs. Otherwise, arcs are simulated using line segments.

Related topics

Print/Plot dialog box

Grid Array tab (Pad Array Generator)

Use this tab to create a Grid array or Ball Grid array.

X Direction area

Number (p) Number of pad columns in the X direction.

Spacing (x) Spacing between column centers in the X direction, entered in the selected units. For example, if you had a 60 mil pad and wanted a 20 mil space between the pad columns, you would enter 80 in this field. Total spacing cannot exceed 32 inches (812.8000 mm).

Start Value The value for the first pad in the array. The start value must be entered in the entry box as a number, and is converted to numeric or alphabetic pad labels. Zero corresponds to the letter “A” when using alphabetic pad labels.

Increment Increment at which pads are labeled. For example, if you wanted to skip from 1 to 3 to 5 and so on, you would enter a value of 2.

Numeric or Alphabetic The Pad Name buttons control this setting. See description below.

Void Cols(vx) Use this entry box to specify the number of interior pins that need to be removed from the center of a BGA pattern. This value should be in-synch with the value in the Void Rows entry box.

Center Cols(cx) Use this entry box to specify the number of interior pins that need to be replaced in the center of a BGA pattern. This value should be in-synch with the value in the Center Rows entry box.

Y Direction area

Number (q) Number of pad rows in the Y direction.

Spacing (y)	Spacing between row centers in the Y direction, entered in the selected units. Total spacing cannot exceed 32 inches (812.8000 mm).
Start Value	The start value for the Y direction label of the grid.
Increment	Increment at which pads are labeled.
Numeric or Alphabetic	The Pad Name buttons control this setting. See description below.
Void Rows(vr)	Use this entry box to specify the number of interior pins that need to be removed from the center of a BGA pattern. This value should be in-synch with the value in the Void Columns entry box.
Center Rows(cr)	Use this entry box to specify the number of interior pins that need to be replaced in the center of a BGA pattern. This value should be in-synch with the value in the Center Columns entry box.

Options area

Display Pad Name	Displays pad names in the Array Preview window. **Note:** Disabling this feature will speed up display refresh rates.
Display Drill	Displays relative drill information for each pad in the Array Preview window.
Silk Screen	Allows you to set the silk screen spacing from the pin center. The Notch check box is also provided in the Silk Screen group box. If you select this check box, the spacing will always be set wrt pin 1.
Place Outline	Allows you to set the spacing of the outline from the pin center.

Pad Name

Alpha / Numeric This sets the X direction label to Numeric and the Y direction to Alphabetic.

Numeric This sets the X direction label to Numeric and disables the Y direction label.

Padstacks area

Default Padstack This sets the default padstack for all pads. Use the Select button to open the Select Padstacks dialog box, and select a padstack from the libraries.

Note: Changing the Default Padstack will always set Pin 1 to the Default padstack. To use a different Pin 1 padstack, set Pin 1 after you have set the default padstack.

Pin 1 Padstack Use this to override the Default Padstack for Pin 1. Use the Select button to open the Select Padstacks dialog box, and select a padstack from the libraries.

Note: You must set the Pin 1 padstack ***after*** you have set the default padstack.

Related topics

Pad Array Generator dialog box

Group Settings dialog box

The Group Settings dialog box appears when you click the Group Settings button in the Route Strategy tab of the SPECCTRA Automatic Router Parameters dialog box.

Use this dialog box to specify the settings for the group numbers you want to use for the pass types you specify in the Route Strategy tab of the SPECCTRA Automatic Router Parameters dialog box.

Group No.

Displays the number of the group.

Loop(s)

Specify the number of loops in which you want to run the route action you specified for the pass type that uses the group.

Via Grid Override

Specify the via grid override settings for the route action specified for the pass type that uses the group.

For example, if the via grid override is 10 x Route Grid, and the route grid you specified in the Router Setup tab is 2 mils, routing will be done at a via grid of 20 mils.

Tip

Vias are a common source of routing congestion and just a handful of vias can completely close a routing channel. For early routing passes, use a via grid that is at least five times the route grid you specified in the Router Setup tab to leave channels open for traces. When the router stalls, execute a second routing pass loop with a smaller via grid until the router stalls again. Continue in this manner until the via grid matches the route grid. Interspersing Clean passes between Route passes can also be helpful.

Halt On Stall

If the halt on stall setting is Yes, SPECCTRA will halt the route action specified for the pass type that uses the group, if the router is no longer making progress. Any remaining loops are simply skipped.

If the halt on stall setting is No, SPECCTRA will continue the route action specified for the pass type that uses the group for the specified number of loops, even if the router is no longer making progress.

IDF to Layout dialog box

Input IDF Board

Enter the path and filename of the IDF (Intermediate Drawing Format) board file (.EMN) you want to translate into Layout. If you choose the Browse button, you can use the Input IDF Board File dialog box to select this file. After you either choose Open or double-click on the IDF board file's icon, Layout automatically creates paths and filenames for all text boxes that require them.

Input IDF Library File

If you have an IDF Board file, but do not have an IDF library file (.EMP), you may leave this field blank.

Note: Mechanical engineers will sometimes create a board outline, height restriction zones, voids, and route keepouts in a tool like Pro/ENGINEER, then transmit that information to the electrical engineer using an IDF Board file. Since there is no component placement data, there will not be an IDF Library file. In this case, simply leave the text box for the Input IDF Library File blank.

Input Technology or Layout File

If you are creating a new Layout file, enter the name of a technology file (.TCH) in this text box. The technology file serves as a design template.

Note: If you are bringing a board that has undergone thermal analysis back into Layout, enter the name of the original Layout board file (.MAX) in the Input Technology or Layout File text box. If you have shifted component placements to meet your heat dissipation goals, the components in your original Layout board file will be moved to their new locations.

Output Layout File

In this text box, you specify the path and filename for the Layout board file (.MAX) the translator creates.

Overwrite existing files

By selecting this option, you allow Layout to overwrite any preexistent files without presenting you with a warning.

Center placed components

The Center placed components option causes the centroid of the mechanical engineer's part to overlay the centroid of the electrical engineer's part. Select this option if the placement origin for critical components, as defined by the mechanical engineer, does not match the placement origin as defined by the electrical engineer. Note, though, that the reference designator used for the part must match the reference designator specified in your netlist. If it does not, edit the part once you have translated the design into Layout format, but before you load your netlist. Once the reference designator matches your netlist, loading the netlist provides the part with the required footprint, silkscreen, pads, and electrical connectivity information.

Ignore all components

Select this option if you want to discard the component placement information in your IDF Library file.

Do not update board outline or voids

Select this option if you changed the board outline in your IDF board file (during thermal modeling, perhaps), but you want to keep the board outline that's in your original Layout file.

Related topics

Layout to IDF dialog box

IDF to Layout command (Import)

Layout to IDF command (Export)

Hatch Pattern dialog box

The Hatch Pattern dialog box is displayed when you choose the Hatch Pattern button from the Edit Obstacle dialog box.

Hatch Pattern

- **Line** - Specifies a set of parallel lines or hatch marks.
- **Cross hatching** - Specifies two sets of intersecting hatch marks.
- **Solid** - Specifies solid copper pour.

Hatch grid

Specifies the hatch grid of the fill area for copper and copper pour, as measured from center of line to center of line.

Hatch rotation

Specifies the angle between the two sets of intersecting hatch marks of a cross hatched copper and copper pour area. The angle must be a multiple of 45 degrees.

Related topics

Edit Obstacle dialog box

Jumper Lengths dialog box

If you want the autorouter to use jumpers while routing a one-sided board, you must select Jumper Layer as the Layer Type on the Edit Layer dialog box and then choose the Jumper Attributes button to display the Jumper Lengths dialog box. The Jumper Lengths dialog box is also available via the Jumper Settings command command from the Options menu.

If a design uses jumpers, you must use a strategy file that allows for this fact. Two such files are provided with Layout in the Data directory. They are JUMPER_H.SF (for horizontal jumpers) and JUMPER_V.SF (for vertical jumpers).

Jumpers are considered footprints, and you can find them in JUMPER.LLB in the Library directory. Each jumper has a specific length; those in JUMPER.LLB range in size from .100 inches to 1.2 inches.

In the Jumper Lengths dialog box, you specify the jumper lengths and the jumper orientations that are allowed on a layer.

Jumper Designator Prefix

This setting defaults to "W," the prefix for jumpers.

Jumper directions

The autorouter places jumpers on the routing layer in either vertical or horizontal fashion, depending upon the option you select in this settings group. If you want to permit both horizontal and vertical placement, select the first setting (Horizontal or Vertical).

Jumper Length

Determines the jumper lengths the autorouter is allowed to create.

Jumper Footprint

Each jumper length specification must be paired with a footprint of the same length. You can find footprints in the Jumper library with the Library Manager.

Related topics

Edit Layer dialog box

Jumper Settings command

Layers Enabled for Routing dialog box

When you choose either the Layer command from the Tool, Net menu or choose the Net Layers button in the Edit Net dialog box, the Layers Enabled for Routing dialog box appears. Using this dialog box, you can specify which layers you can use for routing connections.

A selected check box next to a layer name indicates that the router may place tracks on that layer. In addition, the Design Rule Check command flags any net that is manually routed on a layer that you haven't designated as a routing layer.

This is particularly useful for autorouting clock lines that have to be sandwiched between power or ground planes.

Layers Enabled for Routing

- TOP - Specifies if the TOP layer is enabled for routing.
- BOTTOM - Specifies if the BOTTOM layer is enabled for routing.
- INNER1 - Specifies if the INNER1 layer is enabled for routing.
- INNER2 - Specifies if the INNER2 layer is enabled for routing.

Plane Layers

- GND - Specifies if the GND layer is enabled for routing.
- POWER - Specifies if the POWER layer is enabled for routing.

Related topics

Edit Layer dialog box

Routing

Layout MAX to HYP window

File menu

Open

Choose this command to select a .MAX file for translation. Translation occurs immediately after you either double-click on the icon of a .MAX file or choose the Open button.

Exit

This command closes the Layout MAX to HYP window.

Help menu

About

When you choose the About command, a dialog box displaying the version number and build number of the HyperLynx translator appears.

Related topics

Layout to Hyperlynx command (Export)

Layout to GenCAD dialog box

Input Layout File

Enter the path and filename of the Layout board file (.MAX) you want to translate to GenCAD. If you choose the Browse button and select a .MAX file using the Input Layout MAX File dialog box, Layout automatically displays a name and directory for the GenCAD file you intend to create.

Output GenCAD File

Enter the name and path of the GenCAD file that is to be created.

Overwrite existing files

Select this option if you want to dispense with the warnings Layout normally issues prior to overwriting existing files.

Use library footprints

If you select this option, Layout places one copy of a footprint shape for each family of devices into the GenCAD file. If you

don’t select this option, Layout places one copy of each component shape in the GenCAD file.

Related topics

Layout to GenCAD command (Export)

Layout to GenCAM dialog box

Input Layout File

Enter the path and filename of the Layout board file (.MAX) you want to translate to GenCAM. If you choose the Browse button and select a .MAX file using the Input Layout MAX File dialog box, Layout automatically displays a name and directory for the GenCAM file you intend to create.

Input GenCAM File

Use this option when importing a GenCAM file.

Output GenCAM File

Enter the name and path of the GenCAM file that you want to create.

Overwrite existing files

Select this option if you want to dispense with the warnings Layout normally issues prior to overwriting existing files.

Related topics

Layout to GenCAM command (Export)

Layout to IPC-356 dialog box

Input Layout File

Enter the path and filename of the Layout board file (.MAX) you want to translate to IPC-356. If you choose the Browse button and select a .MAX file using the Input Layout MAX File dialog box, Layout automatically displays a directory and name for the IPC-356 you intend to create.

Output IPC356 Netlist File

Enter the path and filename of the output netlist. The default extension is .NET.

Overwrite existing files

Select this option if you want to dispense with the warnings Layout normally issues prior to overwriting existing files.

Drills inside pads are through-holes

When you select this option, any pad with a drill appears as a through-hole pad in the generated netlist.

Include copper area with netnames

This option forces copper areas assigned to a component-pin as custom pad shapes in the output netlist. Their netnames correspond to the component-pins to which you assigned them.

Output Format

Variable length records

Outputs a netlist with variable length records.

Fixed length records (no linefeed)

Outputs a netlist with fixed length records, but does not include a line feed. This is the default option.

Fixed length records with linefeed

Outputs a netlist with fixed length records, including line feeds.

Related topics

Layout to IPC-356 command (Export)

Validating Gerber connectivity using an IPC-D-356 netlist

Layout to IDF dialog box

Input Layout File

Enter the path and filename of the Layout board file (.MAX) you want to translate to IDF. If you choose the Browse button and select a .MAX file using the Input Layout MAX File dialog box, Layout automatically displays directories and names for the IDF board and library files you intend to create.

Output IDF Board File

Enter the path and filename of the IDF board file (.EMN) you want to create.

Output IDF Library File

Enter the path and filename of the IDF library file (.EMP) you want to create.

Overwrite existing files

Select this option if you want to dispense with the warnings Layout normally issues prior to overwriting existing files.

Add a carriage return to each record

You may have to select this option if the IDF file is not accepted by your modeling software.

Omit mechanical components

You should select this option during thermal analysis.

Omit through-hole pad drills

If thru-hole component information is not needed during thermal analysis, select this option.

Use display units

If your Layout design units are different than your editor display units, select this option.

Use library footprints

Be sure to select this option if you are creating a 3-D model. Three-D modeling software uses the footprint name as an index to access the correct 3-D model for your component, so it is important to put the original library name for the footprint into your IDF.

Board Thickness

If you use a value other than zero, the value is used to determine the thickness (Z-axis) of the board after translation.

Default component height

Any component in your design that does not have a height value receives the value you put in this option.

Related topics

Layout to IDF command (Export)

Layout to ODB++ dialog box

This dialog box appears in the session frame when from the File menu, you point to Export and choose Layout to ODB++.

Input Layout MAX or MIN file

Enter the path and filename of the Layout board file (.MAX) or the OrCAD ASCII format file (.MIN) you want to translate to the Valor ODB++ or XML++ format, or click Browse to select the .MAX or .MIN file.

Output Valor Directory

Enter the path to the directory where you want the Valor ODB++ or XML++ file to be created, or click Browse to select the directory.

Output Valor Jobname

Enter the name of the ODB++ or XML++ job to be created in the output directory.

Layout creates a folder that has the same name as the job name in the output directory. The ODB++ or XML++ output files are located in this folder.

Note: Valor requires that the job name is in lowercase and must not contain spaces. If you enter a job name in uppercase or with spaces, Layout changes it to

lowercase and converts the spaces to underscore characters (_).

Overwrite existing files

Select this check box if you do not want Layout to prompt you before overwriting existing files.

Output copper pours

Select this check box if you intend to use the Valor Trilogy or Enterprise tool to create Gerber files from your ODB++ output files. You should always perform a connectivity check on your output files to verify correct translation of your design.

Output format

- Create ODB++ Output

  Select this option to translate the Layout file to the Valor ODB++ format

- Create XML++ Output

  Select this option to translate the Layout file to Valor XML++ format

Layout creates a folder that has the same name as the ODB++ or XML++ job name in the output directory. The ODB++ or XML++ output files are located under this folder.

Related topics

Layout to ODB++ command (Export)

ODB++ Export

Layout to SPECCTRA dialog box

This dialog box appears in the session frame when from the File menu, you point to Export and choose Layout to SPECCTRA.

Input Layout File

Enter the path and filename of the Layout board file (.MAX) you want to translate to SPECCTRA format. If you choose the Browse button and select a .MAX file using the Input Layout MAX File dialog box, Layout automatically displays a directory and name for the SPECCTRA file you intend to create.

Output SPECCTRA File

Enter the name and path of the SPECCTRA file that you want to create.

Overwrite Existing Files

Select this option if you want to dispense with the warnings Layout normally issues prior to overwriting existing files.

No layer/object spacing rules

Use this option if you don’t want the translation to adhere to layer or object spacing rules.

Create Do File template

Use this option to create a .DO file template.

Related topics

Layout to SPECCTRA command (Export)

Library Conversion Warning dialog box

This dialog box appears when you choose Save after modifying a footprint from an old Layout library. The message “Attempting to save footprint into older version library,” appears at the top of the dialog box.

Save modified library with new name

When you choose this button, a Create New Library dialog box appears, wherein you are able to rename the library of the footprint you modified. By doing so, you avoid the possibility of overwriting the original legacy library.

Link Footprint to Component dialog box

The Link Footprint to Component dialog box displays when you create a new project, and Layout cannot map a PCB footprint in the netlist to a component in its libraries.

Link existing footprint to component

When you choose the Link existing footprint to component button, the Select Footprint dialog box displays for you to locate and select the desired footprint.

Create or modify footprint library

This command is unavailable in Layout v9.0.

Defer remaining edits until completion

Allows you to run AutoECO in batch mode, then check for errors at completion. To do so, choose Defer remaining edits until completion, then choose the OK button.

Note: The dialog box associated with the "Create or modify footprint library" option cannot be launched in Windows 3.1 when the Layout design editor is already running,

as would be the case if you chose New Design from the session frame's Tools menu, or if AutoECO was launched automatically upon loading a board because its underlying .MNL file had changed. However, the dialog box can be launched from Windows 3.1 if AutoECO is run "standalone" from the session frame. All other functions work identically in Windows 3.1 and Windows NT.

Cancel

Defers the current edit, but the next missing footprint causes the dialog box to reappear. If you defer an edit (or edits) using Cancel, you can still match a component to a footprint in Layout. Choose Database Spreadsheets from the Window menu, then choose Components in the Select Data Window dialog box that appears. Double-click on the Components spreadsheet to bring up the Edit Component dialog box, then press the Footprint button to bring up the Select Footprint dialog box.) Note that the "Defer remaining edits until completion" option will skip the supplementary dialog box used with Cancel.

Related topics

AutoECO

Manual Route Strategy dialog box

The Manual Route Strategy dialog box is actually misnamed: all of the settings available to you within this dialog box concern the behavior of Layout's interactive autorouting tools (the Auto Path and Shove Route tools, respectively). To display the Manual Route Strategy dialog box, choose the Manual Route Strategy command from the Options, Route Strategies menu.

Via Cost

By specifying a via cost, you determine the probability with which the autorouting tool suggests the use of a via for a given track path. A high Via Cost decreases the use of vias (a value of 100 for the Via Cost setting prohibits their use). As a general rule, the number of layers a design contains is inversely related to the number of vias the autorouting tool proposes to use.

Retry Cost

The Retry Cost setting determines the number of attempts the autorouting tool makes before it deviates from suggested parameters. Layout derives these parameters from your chosen strategy file and the other settings in the Manual Route Strategy dialog box.

You should use a low Retry Cost value with a high Via Cost value. This enables the autorouting tool to correct tracks that go against the predominant direction of routing and block possible routing paths.

You should use a high Retry Cost value with a low Via Cost value. This limits the number of vias placed by the autorouting tool and encourages it to route around impediments.

Route Limit

The Route Limit value determines the amount of effort the router expends in trying to route a particular connection. A high Route Limit often produces a circuitous and inefficient track that winds around a large area of the board. A low Route Limit produces shorter, more efficient tracks, but there is a higher chance that the autorouting tool will deviate from your suggested parameters. For example, with a low Route Limit, the autorouting tool might insert a via where a slightly more lengthy track would suffice.

The default Route Limit value of 100 rarely needs to be altered.

Attempts

Use the Attempts scroll bar to set the number of times the router will attempt to route a connection. The Attempts value you select has an affect on both completion time and completion percentage. Time constraints are the main reason to limit the Attempts value; a high number of attempts can waste time that the autorouting tool could use to explore other routing options.

Related topics

Edit Place Pass dialog box

Modify Connections dialog box

The Modify Connections dialog box appears when you select either a pin or a net in the design window and choose Properties from the pop-up menu or the Tool, Pin menu. Use this dialog box to assign a net to the selected pin.

The Net name that appears in the dialog box indicates the net that is currently assigned to the pin. You can change the net that is assigned to a pin by entering the new net name in the appropriate area of the dialog box.

Pin

Specifies the currently selected pin.

Net name

Specifies the name of the net.

Save tracks

Saves the routes that are currently attached to the pin.

Allow nodes

Allows nodes to be attached to the pin.

Related topics

Edit Net dialog box

Modify Nets dialog box

Pin Attachment dialog box

Modify Nets dialog box

In the Modify Nets dialog box, you can change the name of a selected net. To display this dialog box, select the Name Net command from pop-up menu after you select a connection with the Select Tool command (Connection).

New net name

Use this drop-down list to specify the new name of the selected net.

Related topics

Edit Net dialog box

Net Selection Criteria dialog box

To display the Net Selection Criteria dialog box, you open the Nets spreadsheet and then choose either the Select from Spreadsheet command from the Tool, Net menu; the Find/Goto command from the Edit menu; or the Select Any command from the pop-up menu.

Net Name

Use this text box to identify and select a net by name.

Group Number

Use this text box to identify the group number of the net in question.

Net Spacing by Layer dialog box

The Net Spacing By Layer dialog box appears when you choose the Spacing By Layer command from the Net submenu (available from the Tool menu). It is also accessible via the Edit Net dialog box by choosing the Net Spacing button.

This dialog box allows you to override the track-to-track spacing parameter set in the Route Spacing spreadsheet. This is useful for performing global impedance matching.

TOP

The value you enter in this text box sets the track-to-track spacing parameter for the TOP layer.

BOTTOM

The value you enter in this text box sets the track-to-track spacing parameter for the BOTTOM layer.

INNER*X (where X is an integer)*

The value you enter in this text box sets the track-to-track spacing parameter for an INNER layer.

Related topics

Edit Net dialog box

Layers Enabled for Routing dialog box

Net Widths by Layer dialog box

Reconnection Type dialog box

Net Widths by Layer dialog box

When you choose the Width By Layer command (available from the Tool menu and its Net submenu), or the Width by Layer button in the Edit Net dialog box, the Net Widths By Layer dialog box appears. This dialog box allows you to set a specific track width for a net in relation to individual layers.

This dialog box allows you to override any track width indicated in the Nets spreadsheet. This is useful for global impedance matching.

TOP

The value you enter in this text box sets a specific track width for the relevant net on the TOP layer.

BOTTOM

The value you enter in this text box sets a specific track width for the relevant net on the BOTTOM layer.

INNER*X (where X is an integer)*

The value you enter in this text box sets a specific track width for the relevant net on an INNER layer.

Related topics

Edit Net dialog box

Layers Enabled for Routing dialog box

Net Widths by Layer dialog box

Reconnection Type dialog box

Net Spacing by Layer dialog box

Obstacle Selection Criteria dialog box

The Obstacle Selection Criteria dialog box is available when you choose the Find button from the toolbar or the Find command from the context sensitive menu while the Obstacle spreadsheet is active.

Obstacle Name

Use this text box to indicate the name of the obstacle you want to select. Use the asterisk (*) and question mark (?) wildcards to qualify your search.

Obstacle Type

Use this drop-down list to indicate the type of the obstacle you want to select.

Obstacle Layer

Use this drop-down list to indicate the layer that the sought-after obstacle appears on.

Related topics

Edit Obstacle dialog box

Component Attachment dialog box

Package Edit dialog box

The Package Edit dialog box appears when you double-click on a cell in the Packages spreadsheet, or when you choose the Properties command from the pop-up menu after selecting a cell in this same spreadsheet. The Packages spreadsheet appears when you choose Packages from the spreadsheet toolbar button's menu.

Pad

The name of the selected pad appears at the top of the dialog box. If you select two or more pads, Layout indicates the number of selected pads.

Package Name

This text box contains the name of the package that corresponds to the selected pad.

Gate Name

The gate name is a simple alphanumeric character that designates which gate each pin belongs to. A package comprises several gates, each gate having a unique name; a gate comprises a number of pins, each sharing the same gate name.

Pin Name

The Pin Name identifies each pin in terms of its electrical characteristics, so that Layout can swap gates correctly. Note in the example below that each gate uses the names "INA," "INB," and "OUTY." This enables Layout to correctly exchange pins between identical gates.

You can display pin names in the design window by way of the Colors spreadsheet.

Each pin within a gate must have a unique identifier--though they all share the same gate name. Each swappable gate must have identical pin names for corresponding pins.

Pin swapping is handled by Pin Group, not by the pin name, so you should not use the same name for both pins (for example, "INPUT") to show that they can be swapped.

If there are any problems with the pin names, you will see the message "Unable to correlate pin names" when trying to do a manual gate swap.

For example, a "74LS01" has 14 pins, which would be labeled as follows:

Pin 1, Pin name: "OUTY"

Pin 2, Pin name: "INA"

Pin 3, Pin name: "INB"

Pin 4, Pin name: "OUTY"

Pin 5, Pin name: "INA"

Pin 6, Pin name: "INB"

Pin 7, Pin name: "GND"

Pin 8, Pin name: "INA"

Pin 9, Pin name: "INB"

Pin 10, Pin name: "OUTY"

Pin 11, Pin name: "INA"

Pin 12, Pin name: "INB"

Pin 13, Pin name: "OUTY"

Pin 14, Pin name: "VCC"

Gate Group

Gate Group is an integer that is used to tell Layout which gates, otherwise identical, cannot be swapped.

In the example below, gates "A," "B," "C," and "D" are identical electrically. Here, gate "A" can be swapped with gate "B" (gate group #1), and gate "C" may be swapped with gate "D" (gate group #2); however, gate "A" cannot be swapped with gate "D." Gate group "0" is a special case that represents a non-swappable gate.

Pin 1 Gate name: "A" Gate group: 1

Pin 2 Gate name: "A" Gate group: 1

Pin 3 Gate name: "A" Gate group: 1

Pin 4 Gate name: "B" Gate group: 1

Pin 5 Gate name: "B" Gate group: 1

Pin 6 Gate name: "B" Gate group: 1

Pin 7 Gate name: "GND" Gate group: 0

Pin 8 Gate name: "C" Gate group: 2

Pin 9 Gate name: "C" Gate group: 2

Pin 10 Gate name: "C" Gate group: 2

Pin 11 Gate name: "D" Gate group: 2

Pin 12 Gate name: "D" Gate group: 2

Pin 13 Gate name: "D" Gate group: 2

Pin 14 Gate name: "VCC" Gate group: 0

Pin Group

Pin Group is an integer used to determine which pins can be swapped.

Any two pins within the same component and gate that belong to the same pin group (for example, Pin Group #1) can be swapped.

Pin swapping is not handled by the pin name, so you should not use the same name for both pins (for example, "INPUT") to show that they can be swapped. Pin names are used only for correlating pins during gate swaps.

If any two pins you wish to swap are not in the same gate and pin group, you will get the message "Pins are not swappable. Override?" when trying to do a manual pin swap. You can answer "OK" to swap the pins anyway, or "Cancel" to abort the operation.

Pin group "0" (zero) is a special case that represents non-swappable pins.

Pin Type

- **Load** - Load means that the pin is part of an ECL net, but is neither a source nor a terminator. Any pin type other than None will result in the affected nets being automatically designated as High speed (daisy-chain) nets. None (ECL) is used for designating standard TTL type pins. Loads are always "daisy-chained" (connected in series). Loads never begin or end ECL nets, unless either the source or the terminator is missing.
- **None** - None means that the pin is not part of an ECL net, and is neither a source, a terminator, nor a Load. None is used for designating standard TTL type pins.
- **Source** - Source means "ECL source pin." This refers to whether or not the pin is part of an ECL net. Any pin type other than None will result in the affected nets being automatically designated as High speed (daisy-chain) nets. None is used for designating standard TTL type pins. Source pins will always be connected to a Load pin, and never to a terminator, unless there are no loads on the net.
- **Terminator** - Terminator means "ECL termination pin." This refers to whether or not the pin is part of an ECL net. Any pin type other than None will result in the affected nets being automatically designated as High speed (daisy-chain) nets. None is used for designating standard TTL type pins. Termination pins will always be connected to a Load pin, and never to a source, unless there are no loads on the net.

Internal gate swap only

With the "Internal gate swap only" flag, you designate that gates of the selected part can be swapped within a particular component only. This, in combination with the Gate Group criteria, gives you complete control over gate swaps.

If you enable the "Internal gate swap only" option, Layout swaps gates only within components, and not between different components.

Pad Array Generator dialog box

The Pad Array Generator dialog box appears when you check the Use Pad Array Generator option in the Create New Footprint dialog box. The Pad Array Generator helps you to quickly and accurately create complex pad placements.

Array style tabs

The Pad Array Generator can create six styles of pad arrays. Select the appropriate style from one of the following tabs at the top of the dialog box. For more information about each style, refer to the help topic for that specific tab.

- **Dual/Quad Inline tab (Pad Array Generator)** – create arrays that are limited to two columns in the X direction.
- **Connector Stagger X tab (Pad Array Generator) –** create arrays that are numbered from left to right, top to bottom.
- **Connector Stagger Y tab (Pad Array Generator)** – create arrays that are numbered from top to bottom, left to right.
- **QFP/Chip Carrier tab (Pad Array Generator)** – create a Quad Flat Pack or Chip Carrier array.
- **Circular tab (Pad Array Generator)** – create an array of pads in a circle.
- **Grid Array tab (Pad Array Generator)** – create a Grid Array or Ball Grid Array.

Choose the style that most accurately represents the pads on your footprint. The resultant pad array can be edited in the library manager after it has been created.

Style Sample button

Press the Style Sample button to open the Style Sample window. This window displays a sample of the currently selected tab style. If you select a different tab while the Sample Style window is open, it will automatically display the sample for that style.

The Style Sample is labeled with lower case letters. These letters match the parameters in the style tabs, and help you to identify how parameter changes affect your pad array. Numbers and upper case letters indicate how the pads are labeled.

In the upper right corner of the Style Sample window the current number of pads is displayed with the pad calculation equation. Notice that the lower case letters of the equation also match the parameters in each style tab.

Note: With some styles, the display changes depending on the values specified in the style tab - specifically the Circle and QFP/Chip Carrier displays. See the Circular tab (Pad Array Generator) and QFP/Chip Carrier tab (Pad Array Generator) help topics for more information.

To close the Style Sample window, press the Style Sample button again.

Array Alphabet button

The Edit Array Alphabet dialog box (Pad Array Generator) appears when you press the Array Alphabet button. This dialog allows you to select the letters of the alphabet that are used to label the pads in your array. The default alphabet is set to the JEDEC standard.

Related topics

Edit Array Alphabet dialog box (Pad Array Generator)

Select Padstack dialog box (Pad Array Generator)

QFP/Chip Carrier tab (Pad Array Generator)

Circular tab (Pad Array Generator)

Dual/Quad Inline tab (Pad Array Generator)

Connector Stagger X tab (Pad Array Generator)

Connector Stagger Y tab (Pad Array Generator)

Pin Attachment dialog box

Pin Attachment

- **None** - Specifies that the obstacle is not associated with a particular pin of the component or footprint.
- **Attach to pin** - Specifies that Layout attaches the obstacle to a particular pin of the component or footprint.
- **Pin Name** - Specifies the name of the pin to which the obstacle is attached.

Related topics

Edit Obstacle dialog box

Component Attachment dialog box

Place Settings dialog box

In the Place Settings dialog box, you can select options for controlling the behavior of the Component command (Place) and Matrix command (Place) placement modes.

To display the Place Settings dialog box, choose the Place Settings command from the Options menu.

Matrix Place Configuration

Allow Outlines to Overlap

If you select this option, the Matrix command (Place) may produce a component placement with overlapping component outlines.

Auto Swap

If you select this option, choosing the Auto, Place, Matrix command may result in some component swapping, in order to ensure the best component placement.

Quick Place Configuration

Fast Reconnect Mode

If you select this option, Layout performs the Quick Place command approximately 3 times faster than it would otherwise and places your components with approximately eighty percent of the accuracy that it achieves without this selected.

Iterations

Use the scroll bar and arrow buttons (or enter a value in the text box) to adjust the number of iterations performed during the Quick Place process. The higher the value, the greater the number of algorithms Layout employs during the process. The default value is 5.

If time is of the essence, set a low Iterations value. A high Iterations value is more time consuming, but it makes an optimal placement of components more likely.

Attempts

Use the scroll bar and arrow buttons (or enter a value in the text box) to adjust the number of iterations performed during the Quick Place process. The higher the value, the greater the number of different placements Layout attempts during each iteration (above). The default value is 5.

If your time is limited, specify a low Attempts value. If you want the best possible chance at an optimal placement of components, set a high Attempts value. A high number of attempts takes a longer time to complete than a low number.

Related topics

Board command (Place)

Matrix command (Place)

Plating Properties dialog box

The Plating Properties dialog box appears when in the Stackup Editor, you first select Top or Bottom check box and then click on the enable button next to these check boxes.

This dialog box is used to specify the foil properties. A foil on the top or the bottom layer of a PCB is characterized either by weight or by the thickness of the coated copper layer.

For example, you can either say that the foil layer has 2 ounces per square foot of copper or that the foil has 0.07mm-thick copper layers.

In the Plating Properties dialog box, specify the thickness of the copper layer in the Plating Thickness text box. Similarly, use the Plating Weight text box to specify the weight of the copper plating.

Post Process Settings dialog box

In the Post Process Settings dialog box, you set the scaling, shifting, device, and filename for all output from Layout, including Gerber, DXF, and any formats available in Print Manager. The dialog box contains the full paths to the output files. Output files, with names in the form board_name.TOP, are created for each layer.

To open the Post Process Settings dialog box

1. From the Options menu choose Post Process Settings. The Post Process spreadsheet appears.
2. Click a layer name to select that layer.
3. Right click on the layer name and choose Properties from the pop-up menu. The Post Process Settings dialog box appears.

Output

Format

Gerber RS-274D

A file format that can be read by Gerber and other photoplotter systems that require separately or previously defined aperture lists.

Extended Gerber

When this option is selected, Layout uses the Gerber 274X standard, which includes aperture shapes directly in the output file for each artwork layer. If you disable this option, Layout's Gerber output uses the 274D standard, which requires a separate aperture list. The aperture list is generated in GerbTool format, so that it can be read directly into Layout's Gerber viewer.

DXF

DXF is a graphics format created by AutoCAD. Select this option if you want to convert the layers you highlighted in the Post Process spreadsheet to the DXF file format.

Print Manager

This option permits you to send your board design to a printer. If you want to use HPGL, or a laser or dot matrix printer, you must set configuration options in the Windows Print Manager. For more information about the Print Manager, see your Windows documentation

Options

Keep Drill Holes Open

With this option selected, drill holes appear as open areas (they appear in the background color). This option is available for printer and Extended Gerber output.

Create Drill Files

When you select this command from the Post Processing menu, Layout produces a set of drill tape files in Excellon format and places them in the design directory. One file is created for each drill layer pair. The file naming convention is Layer#_Layer#.TAP, so that vias on and between layers 3 and 4 are in the file 3_4.TAP, for example. The plated through-hole drills are in the file THRUHOLE.TAP and the non-plated through-hole drills are in the file THRUHOLE.TAP

During the manufacturing process, the drilling machine reads this set of files to determine the size and location of the drill holes in the board.

Unless you shift the output along either the horizontal or vertical axis (see below), the drill tape coordinates match the coordinates that you see in the design window.

Overwrite Existing Files

If you select this option, Layout writes over any existing files with the same filename and extension.

Enable For Post Processing

This option allows you to post process the layer or layers you selected in the Post Process spreadsheet. If a layer(s) is enabled for post processing, the appropriate cell under the Batch Enabled column on the Post Process spreadsheet reads “Yes.”

File Name

Use this text box to assign a filename to your output.

Output Settings

Plot Title

The text you enter in this text box defines your plot.

X Shift

The value you enter in this text box is the horizontal distance the plot shifts relative to the board datum. This text box is disabled if you select the Center on Page option (see below).

Y Shift

The value you enter in this text box is the horizontal distance the plot shifts relative to the board datum. This text box is disabled if you select the Center on Page option (see below).

Center on Page

Selecting this option centers your board on the printed page. The Center on Page option is selected by default, which means that the drill tape coordinates match the coordinates that you see in the design window.

Mirror

With this option selected, the output image flips along the horizontal axis. Only one image of the board or selected layers appears

Scale Ratio

These text boxes allow you to enter a ratio for the scaling of output. If, for example, you enter “2” in the first box and “1” in the second, output appears twice as large as the original image. Reversing these values produces an image half the size of the original.

Rotation (CCW)

These options rotate the output image counter-clockwise by the specified number of degrees.

Note: Customers have reported that according to the Product Comparison Guide for HP-GL/2 and HP RTL Peripherals published by Hewlett-Packard, the following devices are obsolete and no longer supported. Therefore, you may be unable to acquire a current, working driver for the following output devices.

- HP DesignJet plotters (except 200, 600, and 650C)
- HP PaintWriter XL printer
- HP DraftMaster Series plotters - SX, RX, and MX (except the SXplus, RXplus, and Mxplus)
- HP 7600 Series plotters - Models 240D/E, 250, 255, and 355

Certain companies (for example, WinLINE) produce drivers. You can contact WinLINE on the internet at http://www.winline.com.

Combine Plated/Non-Plated Thru Holes

By default, Layout outputs drill tape files named THRUHOLE.TAP for plated through-holes and THRUHOLE.NPT for non-plated through-holes in your board.

Select this check box if you want Layout to output a single drill tape file named THRUHOLE.TAP for both plated and non-plated through holes.

Related topics

Gerber Preferences dialog box

Print/Plot dialog box

Edit Apertures dialog box

Print Catalog dialog box

File Name

- Input Catalog File - enter the path and file name of the catalog file to print. Only select a MAX file that has been created as a catalog file.

Options

- **Force Black and White** - choose this option to print all elements in black on a white background.
- **Center on Page** - centers the catalog page on the printed page.
- **X Shift and Y Shift** - when the Center on Page option is not selected, use these options to set the

distance of the catalog page from the lower left corner of the printed page.

- **Fit to Page** - this option stretches the catalog page to the printed page. The original aspect ratio of the catalog page is maintained.

  Note: To create pages that fit into a three ring binder, check Fit to Page and use an appropriate X Shift to move the image away from the ring holes.

- **Scaling** - when the Fit to Page option is not selected, enter a percentage to scale the output up or down.

- **Print to File** - choose this option to send the output to a PRN file.

Note: If you use the Browse button to locate the input catalog file, the Print to File edit box is preloaded with a default catalogname.PRN. Even though the filename is preloaded, a PRN file is only created if you select the Print to File option.

Print/Plot dialog box

To display the Print/Plot dialog box, activate the design window then choose Print/Plot from the File menu.

In the Print/Plot dialog box, Layout gives you the option of sending output to a printer, sending output to a print file (.prn) or sending output to a DXF file (.dxf).

To print in Layout, you must install a printer by using the Windows setup program (refer to your Windows documentation for further information on installing and adding printers). In Layout, both printing and plotting are governed by the Windows Print Manager. The image printed follows the settings of the Color spreadsheet.

To view and modify the content of printed output

1 From the Options menu, choose Post Process Settings. The Post Process spreadsheet appears.

2 Position the pointer over a row you would like to print and choose Preview from the pop-up menu. The background of the design window changes to white and the layer you chose to preview is displayed. Choose Tile from the Windows menu.

3 If a layer or object did not appear in the design window when you chose Preview, choose the color toolbar button and select the row under the Color column that corresponds to the layer you would like to have included in your print out.

4 Choose a color from the pop-up menu. The layer whose color you changed appears in the print preview.

5 To restore the design window, right-click on the Post Process spreadsheet and choose Restore Original Colors from the pop-up menu.

Print/Plot Format

Title

Text string that defines your plot, used as comment only.

DXF

Output is in the form of a DXF file (.dxf).

Print Manager

Prints the entire drawing at the scale specified in the Scale Ratio text box and uses as many pages as are necessary. If you want to use HPGL, or a laser or dot matrix printer, you must set configuration options in the Windows Print Manager. For more information about the Print Manager, see your Windows documentation.

Keep drill holes open

Causes the drill holes to display as open areas. This option is only supported when printing to a printer.

PrintPlot Current View

Prints the active window and fits the image onto a single sheet. If this option is selected, the Print/Plot Settings have no effect on output. If the active window is a board, it prints at the current zoom setting.

Print/Plot To File

Output is in the form of a print file (.prn).

File Name

Activates when you select Print/Plot to File, or DXF. You can then name your file.

Print/Plot Settings

Shift (X and Y)

Shifts the plot relative to the board datum by X and Y distance. Activate these options by clearing the Center on Page check box.

Center on Page

This option centers the output image on the printed page. The Center option is the default option, which means that the drill tape coordinates match the coordinates that you see in the design window.

Mirror

With this option selected, the output image flips along the horizontal axis. Only one image of the board or selected layers appears

Scale Ratio

These text boxes allow you to enter a ratio for the scaling of output. If, for example, you enter “2” in the first box and “1” in the second, output appears twice as large as the original image. Reversing these values produces an image half the size of the original.

Rotation (CCW)

These options rotate the output image counter-clockwise by the specified number of degrees.

Note: You do not need to specify a plotter driver in Layout when the print manager is the selected output device, because Layout uses the plotter driver that you have selected as your Windows plotter. For additional information on plotters and drivers, see your Windows documentation.

If the print manager is not the selected output device, then you need to select and output a device supplied by Layout (such as Gerber or DXF).

Note: Customers have reported that according to the Product Comparison Guide for HP-GL/2 and HP RTL Peripherals published by Hewlett-Packard, the following devices are obsolete and no longer supported. Therefore, you may be unable to acquire a current, working driver for the following output devices.

- HP DesignJet plotters (except 200, 600, and 650C)
- HP PaintWriter XL printer
- HP DraftMaster Series plotters - SX, RX, and MX (except the SXplus, RXplus, and Mxplus)

- HP 7600 Series plotters - Models 240D/E, 250, 255, and 355

Certain companies (for example, WinLINE) produce drivers. You can contact WinLINE on the internet at http://www.winline.com.

Note: Windows users need to be sure that their printer is listed as active, and is the default, or Layout will not be able to access it. If these fields are not set properly, you will receive the message "Failed to Init Printer."

Note: To print a Query window, first save the information to a file using Output from the Edit menu. You can then print the file using the standard Print option from the File menu in the Windows File Manager.

Related topics

Gerber Preferences dialog box

Edit Apertures dialog box

Post Process Settings dialog box

Properties (Layer) dialog box

This dialog box is invoked when you click the Properties button located within the Cross Section View section of the Stackup Editor.

This dialog box enables you to specify the properties of the selected layer.

Property..	Used to..
Name	Specify a name for the selected layer.
Material	Specify whether copper or plated copper will be used for manufacturing the selected layer.

Property..	Used to..
Thickness	Enter the thickness of the selected layer. This is in default units used in the Layout board and are specified in the System Settings dialog box
Electrical conductivity	Specify the conductivity of the selected layer in `mho/cm`.
Film Type	Specify whether the postprocessing output should be in positive or negative.
Pullback	Specify the distance between the copper and the edge of the plane layer. **Note:** Pullback is specified only in case of power or ground planes.

Properties (Solder Paste) dialog box

This dialog box is invoked when you click the button next to the Solder Paste check box in the Stackup Editor. In the Properties dialog box you can specify the properties of the solder mask to be applied.

a. Select the Change solder paste pad size check box.

b. Specify the pad size in the enabled text box.

Tip

When you specify the solder paste pad size, padstack information for that layer, such as pad height and pad width, may get modified. This change is reflected in the Padstacks spreadsheet.

Properties (Solder Mask) dialog box

Stackup Editor supports following masking material:

- Dry

  Dry film is useful in tenting over holes or when three dimensional circuit structures are involved.

- Wet

  A wet solder mask is a plastic layer that resists wetting by solder and keeps islands of solder from running together. It also protects the outside conductors layers from abrasion and corrosion.

QFP/Chip Carrier tab (Pad Array Generator)

Use this tab to create a Quad Flat Pack or Chip Carrier array.

X Direction area

Number (p) Number of pads in the X direction.

Spacing (x) Spacing between pad centers in the X direction, entered in the selected units. For example, if you had a 60 mil pad and wanted a 20 mil space between the pads, you would enter 80 in this field. Total spacing cannot exceed 32 inches (812.8000 mm).

Start Value The value for the first pad in the array. The start value must be entered in the entry box as a number, and is converted to numeric or alphabetic pad labels. Zero corresponds to the letter “A” when using alphabetic pad labels.

Increment	Increment at which pads are labeled. For example, if you wanted to skip from 1 to 3 to 5 and so on, you would enter a value of 2.
Numeric or Alphabetic	Sets alphabetic or numeric labeling style. When using Alphabetic labeling, you can specify what letters to use in the Array Alphabet dialog.

Y Direction area

Number (q)	Number of pads in the Y direction.
Spacing (y)	Spacing between pad centers in the Y direction, entered in the selected units. Total spacing cannot exceed 32 inches (812.8000 mm).
Start Value	Disabled
Increment	Disabled
Numeric or Alphabetic	Disabled

Options area

Stagger (w)	Distance that the outer pads of the X row are from the sides of the array. The allowed range is –10.0000 in. (-254.0000 mm) to 10.0000 in. (254.0000 mm).
Stagger (z)	Distance that the outer pads of the Y row are from the top and bottom of the array. The allowed range is –10.0000 in. (-254.0000 mm) to 10.0000 in. (254.0000 mm).

Display Pad Name

Displays pad names in the Array Preview window.

Note: Disabling this feature will speed up display refresh rates.

Display Drill

Displays relative drill information for each pad in the Array Preview window.

Silk Screen

Allows you to set the silk screen spacing from the center.

The Notch check box is also provided in the Silk Screen group box. If you select this check box, the spacing will always be set wrt pin 1.

Place Outline

Allows you to set the spacing of the outline from the pin center

Pin1 area

Corner (QFP)

Sets the location of Pin 1 at the corner of the pad array. Use this option to create Quad Flat Packs. When you change this option, the Style Sample window display changes with your selection.

Center (CC)

Sets location of Pin 1 at the top center of the array. Use this option to create Chip Carriers. When you change this option, the Style Sample window display changes with your selection.

Padstacks area

Default Padstack

This sets the default padstack for all pads. Use the Select button to open the Select Padstacks dialog box, and select a padstack from the libraries.

Note: Changing the Default Padstack will always set Pin 1 to the Default padstack. To use a different Pin 1 padstack, set Pin 1 after you have set the default padstack.

Pin 1 Padstack

Use this to override the Default Padstack for Pin 1. Use the Select button to open the Select Padstacks dialog box, and select a padstack from the libraries.

Note: You must set the Pin 1 padstack *after* you have set the default padstack.

Related topics

Pad Array Generator dialog box

Reconnection Type dialog box

When you select nets from the Nets spreadsheet and select Properties from the popup menu, the Edit Net dialog box appears. Click the Net Reconn button to edit the reconnection rules for each type of reconnection recognized by Layout.

None

Prohibits the router from reconnecting any point-to-point connections for the selected net. This option is used for critical nets that must be connected in a certain predetermined order. This disables reconnection in both the autorouter and in the option Edit/Reconnect Nets. In addition, nets are routed in the order of their appearance in the netlist.

Vertical

Causes the router to seek primarily vertical paths for each connection within a net. Used for VCC and ground.

Horizontal

Causes the router to seek primarily horizontal paths for each connection within a net. This is generally used for VCC and ground.

Std. Orthog.

The Standard Orthogonal option orders the router to seek the easiest path between any two points within a net (usually the shortest distance, but with a predisposition for horizontal or vertical routes where available). This is the default, and should be used for all routing of standard digital signals.

High speed

Prohibits "T" connections, and causes the router to daisy-chain the connections in the net from the Source to the load, and then to the Terminator rather than creating tracks to find the shortest route. This is most often used with high speed parts such as ECL technology. It is often enabled in conjunction with disabling share on critical nets.

No Dyn. Reconn

By default, Layout uses dynamic reconnect, which is a method of calculating where the closest pin belonging to the same net you're routing is, then redrawing the ratsnets line to connect to the closest pin. The No Dyn. Reconn option disables dynamic reconnect, with the result that you don't have to wait for Layout's ratsnets calculations and redrawing. Because of this, selecting No Dyn. Reconn is especially useful when routing large nets. Note that No Dyn. Reconn is not available for use with the None or High speed types of reconnection.

Note: You must have source and terminator pins assigned in the Package Edit dialog box in order to use High speed for automatic ECL routing. Without these assignments, the router will daisy-chain the routes, but it will use an arbitrary source and terminator.

Related topics

Edit Net dialog box

Layers Enabled for Routing dialog box

Net Widths by Layer dialog box

Net Spacing by Layer dialog box

Rename Direction dialog box

The Rename Direction dialog box is available when you choose the Components Renaming command from the Options menu. With this dialog box, you can configure the behavior of the automatic renaming process. You perform automatic renaming by choosing the Rename Components command from the Auto menu.

The Rename Direction dialog box contains eight options; each option is a strategy used to specify the directional flow of component renaming. If you select the Right, Down option, Layout begins at the upper left-hand corner of the board, renames components from left to right across your board, and then moves down, again renaming components from left to right. This pattern of action is repeated for the entire board.

Note that choosing the OK button in this dialog box does not rename components. By choosing OK, you are simply affirming your selection of a renaming strategy (directional option). You must choose the Rename Components command from the Auto menu to implement this strategy and rename your components.

Related topics

Components Renaming command

Rename Components command

Replace dialog box

The Replace dialog box appears when you attempt to change the footprint of a current component on the board.

If the new footprint has the same name as a footprint currently in use, then you are given the choice to either:

- **Replace footprint for all components -** This option updates all components that use the current local footprint to the newly selected footprint in the external library.

or

- **Replace footprint for selected components -** This option updates only the footprint of the currently selected component. If you choose this option, Layout will append an underscore and a number to the footprint name. For example, SM/R_0805 becomes SM/R_0805_1. This allows the new footprint to be unique, while other components that use the originally named footprint remain unchanged.

Replace Footprint dialog box

The Replace Footprint dialog box appears when you click the Replace Selected Footprint button in the Configure Design Library dialog box.

You can use this dialog box to search for the required footprint and replace the current footprint selection for the part in the Configure Design Library dialog box with the required footprint.

Name

Select this check box if you want to search footprints by name.

Enter the name of the footprint you want to search for. You can use the * (asterisk) character to perform a wildcard search.

Match whole word only

Select this check box if you want to find only footprints that match the full footprint name you specified.

For example, if you enter the footprint name as VRES1 and do not select this check box, footprints with names such as VRES1, VRES10, VRES11, and so on will be found. If you select this check box, the search will only find the VRES1 footprint.

Min. pin count

Select this check box if you want to find footprints that have at least the specified number of pins and also have the same pin names as the pin names on the part in your Capture schematic.

Enter the minimum pin count.

Max. pin count

Select this check box if you want to find footprints that do not have more than the specified number of pins and also have the same pin names as the pin names on the part in your Capture schematic.

Enter the maximum pin count.

Search in

Configured libraries

Select this check box if you want to search the libraries you have made available for use in Layout. For more information on making a library available for use in Layout, see Making libraries available for use on page 531.

Other libraries

Select this check box if you want to search the libraries you have not made available for use in Layout.

Click the browse button to select the libraries.

Note: You cannot select libraries existing in multiple directories. Copy all your custom libraries to a single directory.

Search

Click this button to search for footprints that meet the specified search criteria.

Search Results

The footprints that match the search criteria are displayed in the Footprints list. The libraries in which the footprints exist are displayed in the Libraries list.

Libraries

Select a library to display the footprints in the library that match the search criteria.

Footprints

Displays the footprints that match the search criteria.

Replace

Select the required footprint in the Footprints list and click this button. The current footprint selection for the part in the Configure Design Library dialog box is replaced with this footprint.

Related topics

Creating design libraries

Replace Footprint (footprint name) dialog box

The Replace Footprint dialog box appears when you double-click on the row for a part for which multiple footprints were found in the Configure Design Library dialog box.

You can use this dialog box to change the footprint for a part for which multiple footprints were found.

Available footprints for the symbol

Displays all the footprints available for the part.

Replace

Select the required footprint and click this button. The current footprint selection for the part in the Configure Design Library dialog box is replaced with this footprint.

Related topics

Creating design libraries

Route Settings dialog box

The Route Settings dialog box is the primary means of control for all routing tools in Layout. It is divided into three groups:

Route Mode, Interactive Auto Route Settings and Manual Route Settings.

Route Mode

When you first start routing, you should choose one of the four route modes available; choosing a route mode disables options that are not applicable to the mode you selected. The top two modes are for manual routing—the bottom two concern autorouting.

Add/Edit Route Mode

This mode lets you select individual connections (using the left mouse button) for non-shove, manual routing of critical nets, and minute movements of pre-existing tracks and vertices. This routing option may be used with DRC enabled or disabled.

To create a vertex, either use the left mouse button or press SPACEBAR on your keyboard. To finish a track in Add/Edit route mode, either press SPACEBAR or click the left mouse button on the destination pad. You can also finish routing a track by choosing Finish from the pop-up menu. Manual Route is generally for those situations when you need more control over routing than is available with Auto Path.

You can also enter Add/Edit route mode by choosing its toolbar button or by choosing Track, Select Tool from the Tool menu.

Edit Segment Mode

You can enter Edit Segment mode several ways: the Edit Segment toolbar button, the Track Segment, Select Tool command from the Tool menu, the S key on your keyboard or the Segment command on the pop-up menu. This last method of access is available only while the Add/Edit manual route tool is in use.

Shove Track Mode

This mode automatically moves or “shoves” existing tracks to make room for another connection that you’re routing. With the Shove Route tool, you can shove and route individual connections and manually route critical tracks and vertices.

In order to use Shove Route, the environment must have DRC activated in the User Preferences dialog box.

Use Shove Route when you want the power of the shove feature and greater control over routing than is available with the Auto Path tool.

Use the left mouse button to select individual connections. Move the pointer and use either the left mouse button or SPACEBAR to create a vertex; this vertex can be on or near an existing track. Existing tracks will move out of your way after you create the vertex. To finish a track in Shove Track route mode, move the pointer over the destination pad and either press SPACEBAR or click the left mouse button. If there is only a short amount of unrouted connection remaining in a given track, then you can finish routing by choosing Finish from the pop-up menu.

High Power, Medium Power, Low Power

Select one of these three options to specify the level of force employed by the Shove Route tool when moving existing tracks. The High Power setting permits the Shove Route tool to rip-up, shove and reroute tracks, the Medium Power setting allows the tool to shove tracks and push them around other items and the Low Power setting permits the tool to move existing tracks only slightly.

In addition to being available via the Route Settings dialog box, the Shove Route tool can also be activated by choosing the Shove Route toolbar button.

Auto Path Route Mode

When you use the Auto Path tool, the router displays a possible track when you select a ratsnest or pin. As you move the cursor, the suggested track changes position. When you click the left mouse button, the Auto Path tool places the track as it appears using the shove ability of the Shove Route tool (see above), thereby clearing away any imposing tracks. In order to use Auto Path, DRC must be enabled in the User Preferences dialog box.

Auto Path is useful for routing dense board layers, especially if the route you are working on must go against the predominant direction of existing tracks.

Suggest Vias

When you use the Auto Path tool with the Suggest Vias option selected, Layout displays potential via locations. If the Suggest Vias check box is cleared, then you must place vias yourself. This can be done by first choosing a destination layer using the layer pull-down menu or the appropriate hotkey. There are then several ways to add a vias or a free via by pressing V or E, respectively, or by choosing either the Add Via command or Add Free Via command commands from the pop-up menu.

Interactive Auto Route Settings

Allow Off-Grid Routing

This option allows the Auto Path to display possible routing paths without regard to the routing grid. Enabling this option is the only way to permit the Auto Path tool to end tracks at an obscure angle of approach.

Shove Components

This option permits Auto Path to shove components in much the same way as it shoves tracks. That is, when you place a

vertex using the left mouse button or SPACEBAR, any imposing components are moved away from this vertex (unless these components are locked in position).

Maximize 45 Corners

Selecting this option lets the Auto Path tool optimize routing space with vertices of 45 or 90 degrees. With this option cleared, the autorouter creates 90-degree corners only.

Use All Via Types

This permits Auto Path to route a connection with the optimal via type from those vias that are defined in the Padstacks spreadsheet. Auto Path only uses vias when necessary and does not choose an optimal via if you specifically assign one to a net. If this option is not selected and you have not specified a via for use with a given net, then Layout uses VIA1 (the default via type).

Manual Route Settings

Snap to Grid Routing

With this option selected, the segment that you are routing moves from grid point to grid point, so that you cannot create a track off of the routing grid. When you clear this check box, you are able to route regardless of the track’s relationship to the routing grid.

Use Routing Hints

Routing hints allows you to give the manual routing tool a directional "hint" for proceeding from a pad, via, or vertex.

Never	The router does not use directional hints. It uses its default algorithm.

Pads/Vias Only	The router uses directional hints only when exiting pads and vias.
Always	The router uses directional hints when exiting pads, vias, and from the last routed segment endpoint.

Drawing Method

The four options in the Drawing Method settings group concern the angle of vertices created during manual routing. The Any Angle Corners option allows you to create a vertex of any possible angle; the connection segment attached to the Add/Edit routing tool's crosshairs rotates freely through 360 degrees when this option is selected. The 45 Corners setting allows you to create vertices of 45, 90 and 135 degrees while you route. The 90 Corners setting restricts vertices to 90-degree angles. The Curve Corners option gives you the ability to place curved tracks on your board while you route manually. With the Add/Edit routing tool selected, you can create curved, horizontal and vertical tracks (you cannot readily create 45 degree vertices with this option selected).

Related topics

Select Tool command (Track)

Select Tool command (Track Segment)

Manual Route Strategy dialog box

SeedVia tab (Preferences)

Distance between the broken connections

SPECCTRA adds a via for each connection that exceeds the specified length in both the X and Y directions. The value is in design units.

The default is 1000.

Place Vias Under SMD Components

Select this check box if you want vias to be added under SMD components when you route a design with two signal layers.

Miter Corners tab (Preferences)

Miter Passes

Specify the number of passes of the miter command that you want SPECCTRA to run. The default is 4 passes.

Note: The miter command automatically stops once minimum spacing is exceeded.

Miter Pin and Via Exits

Select this check box if you want to change 90-degree wire corners to 45 degrees for wires exiting pins and vias.

Slant Wrong-way Segments

Select this check box if you want to replace segments running against the specified routing direction on a layer with 45-degree segments.

Miter T Junctions

Select this check box if you want to miter wires meeting at a T junction.

Miter at Bends

Select this check box if you want to change 90-degree wire corners to 45 degrees.

Route Strategy tab (SPECCTRA Automatic Router Parameters)

Strategy

Specify Routing Passes

Select this option if you want to route your design using SPECCTRA by running route passes in a specific sequence.

Specify the options for running route passes in Specify Routing Passes.

For information on how to specify the routing passes you want to run in a specific sequence, see Routing Passes Example on page 807.

Use Smart Router

Select this option if you want to route your design using the smart routing option in SPECCTRA, which automatically routes and executes commands based on an evaluation of your design.

When you use the smart routing option, SPECCTRA adjusts the autorouting strategy based on the conflict reduction rate, the routing completion rate, the failure rate, and the number of layers. It applies bus routing, if necessary, and runs clean passes after all connections are completed.

Specify the smart routing options in the Use Smart Router tab.

Specify Routing Passes

The following options are available when you select the Specify Routing Passes option.

Preroute and Route

Lets you specify the routing actions which you want to be performed, in a specific sequence. You can also specify that a routing action should be performed using conditional statements.

For more information on how to specify the routing passes you want to run in a specific sequence, see Routing Passes Example on page 807.

Group Settings

Click this button to display the Group Settings dialog box. Here you can specify the number of loops, via grid override settings, and halt on stall settings for the group number you want to use for a pass type.

Gr. No.

Click on the drop-down list to select the group number you want to use for a pass type.

Selecting a group number for a pass type lets you run the routing action specified for the pass type using conditional statements.

Note: Select the same group number for pass types whose route actions you want to run together. For example, if you want to run Filter and Route as part of the number of loops specified for a group number, select the same group number for the Filter and Route pass types.

Pass Type

Specifies the routing action that is performed. Click on the drop-down list to select a pass type.

Select:

- Fanout, to route short escape wires from pads to vias.
- Seed Vias, to break a single diagonal connection into two shorter connections by adding a via.

  Click the Preferences button to display the Preferences dialog box and specify the options for the Seed Via pass in the SeedVia tab of the Preferences dialog box.
- Route, to route the design.
- Clean, to improve manufacturability by removing vias and bend points and by changing SMD entries and exits.
- Testpoint, to assigns test points to signal nets.
- Critic, to eliminate acute angles and remove extra bends in wires without performing rip-up and reroute operations.
- Filter, to remove final routing conflicts by executing route passes that increase the conflict cost and minimize the number of unconnected wires. You can then route the unconnected wires manually.

  Filter is typically used to finish routing a design that has stopped converging. This should be your last resort when trying to achieve routing completion.

Note: The pass types will be performed in the order in which they are listed in the Route Strategy tab.

Passes

Specifies the number of passes for the pass type. You can specify the number of passes for the following pass types:

- Fanout
- Route
- Clean
- Filter

SPECCTRA uses the number of route passes you specify as long as conflicts remain or connections are unrouted. Once wiring is 100 percent complete with no crossing or clearance violations, unused route passes are skipped. If there are crosstalk or maximum and minimum length violations, route passes continue until these violations are also resolved.

Note: A minimum of 25 route passes is suggested for the initial series of routing passes. After these initial 25 routing passes, you should run two clean passes by using the clean command. The clean command rips-up and reroutes every connection, removes unnecessary vias and bends, and alters the routing problem by making new or different routing channels available for the next series of route passes. You will see a noticeable improvement in the routing quality after the clean passes.

Start

Specify the starting pass number in the SPECCTRA autorouter cost table. The starting pass number allows you to restart routing where you left off in a previous session. SPECCTRA uses this number to start the series of route passes.

Notes:

- If you do not specify the starting pass number, SPECCTRA calculates the starting pass number based on the completion level of the routing. After the first series of 25 route passes, this value is usually set to 16.
- Do not specify the starting pass number unless you are an experienced SPECCTRA user.

Typical values for the starting pass number are:

- 1: SPECCTRA uses the costing that is applied in the initial routing of a design.
- 6: SPECCTRA uses the costing that is used after the initial five route passes.
  The cost of conflicts is relatively low at this point in the cost table.
- 11: SPECCTRA uses the costing that is used after the initial 10 route passes.
  The cost of conflicts is moderate at this point in the cost table.
- 16: SPECCTRA uses the costing that is used after the initial 15 route passes.
  The cost of conflicts is relatively high at this point in the cost table.

Loop(s)

Displays the number of loops specified for the group number you selected for the pass type.

For example, if the number of loops is 4, the route action you specified for the pass type will execute four times.

Via Grid Override

Displays the via grid override setting for the group number you selected for the pass type.

For example, if the via grid override is 10 x Route Grid, and the route grid you specified in the Router Setup tab is 2 mils, routing will be done at a via grid of 20 mils.

Tip

Vias are a common source of routing congestion and just a handful of vias can completely close a routing channel. For early routing passes, use a via grid that is at least five times the route grid you specified in the Router Setup tab to leave channels open for traces. When the router stalls, execute a second routing pass loop with a smaller via grid until the router stalls again. Continue in this manner until the via grid matches the route grid. Interspersing Clean passes between Route passes can also be helpful.

Halt on Stall

Displays the halt on stall setting for the group number you selected for the pass type.

If the halt on stall setting is Yes, SPECCTRA will halt the route action specified for the pass type that uses the group, if the router is no longer making progress. Any remaining loops are simply skipped.

If the halt on stall setting is No, SPECCTRA will continue the route action specified for the pass type that uses the group for the specified number of loops, even if the router is no longer making progress.

Post Route

The following options are available for controlling post-route actions.

Critic

Select this check box if you want to eliminate acute angles and remove extra bends in wires without performing rip-up and reroute operations.

The following figures show how the critic command improves routing.

Figure 18-1 Notch and bend point removal

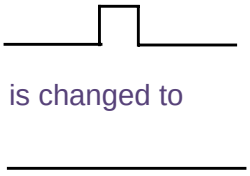


Figure 18-2 Stairstep removal

Filter Routing Passes

Select this check box if you want SPECCTRA to remove final routing conflicts by executing filter route passes that increase the conflict cost and minimize the number of unconnected wires.

Specify the number of filter route passes you want the SPECCTRA to run in the text box.

Note: SPECCTRA will run a maximum of five filter route passes, even if you enter six or more in the text box.

If a few conflicts remain after a large number of route and clean passes are completed, you might want to remove the conflicts and route the remaining connections manually within Layout. You can use this option to remove the conflicts and create unroutes which will display in Layout as rats.

When you specify more than one pass, each pass progressively increases the cost of routing conflicts. During the last filter pass, conflicts are prohibited and conflict-free routing is assured.

Miter Corners

Select this check box if you want SPECCTRA to change 90 degree wire corners at pin and via exits in your board to 45 degrees.

By default SPECCTRA runs four passes of the miter command in an attempt to converge on efficient spacing between adjacent mitered corner trace.

Click the Preferences button to display the Preferences dialog box and specify the options for the miter command in the Miter Corners tab of the Preferences dialog box.

Note: The miter command automatically stops once minimum spacing is exceeded in your board.

Delete Conflicts

Select this check box if you want SPECCTRA to delete connections involved in conflicts or violation of high-speed rules, such as length or delay rules.

All routed wires that intersect other routed wires, or violate clearance rules are deleted. Starting with the wires that cause the most intersections and high-speed rule violations, SPECCTRA removes each wire and re-evaluates the violation list.

SPECCTRA deletes one of the two wires (pin-to-pin connections) involved in each conflict.

Note: Do not use the Delete Conflicts option when there are a large number of conflicts in your board. Instead, use the Filter Routing Passes option to remove conflicts.

Preferences

Click this button to specify the options for running the seed via and miter commands.

Use Smart Router tab

The following options are available when you select the Use Smart Router option.

Grid

Minimum Via Grid

Specify the minimum via grid.

If you do not specify the via grid, the via grid settings in the Router Setup tab will be used.

Minimum Route Grid

Specify the minimum route grid.

If you do not specify the route grid, the route grid settings in the Router Setup tab will be used.

Fanout

Fanout if Appropriate

Routes short escape wires from SMD pads to vias if there are more than two signal layers or if the top or bottom layer is not selected for routing.

If this check box is selected, the following options are available.

Via Sharing

Controls whether SPECCTRA allows sharing vias between SMD pads on the same net.

Pin Sharing

Controls whether SPECCTRA can escape to through-pins.

SPECCTRA will escape to a through-pin if the cost is lower than the cost to use a via.

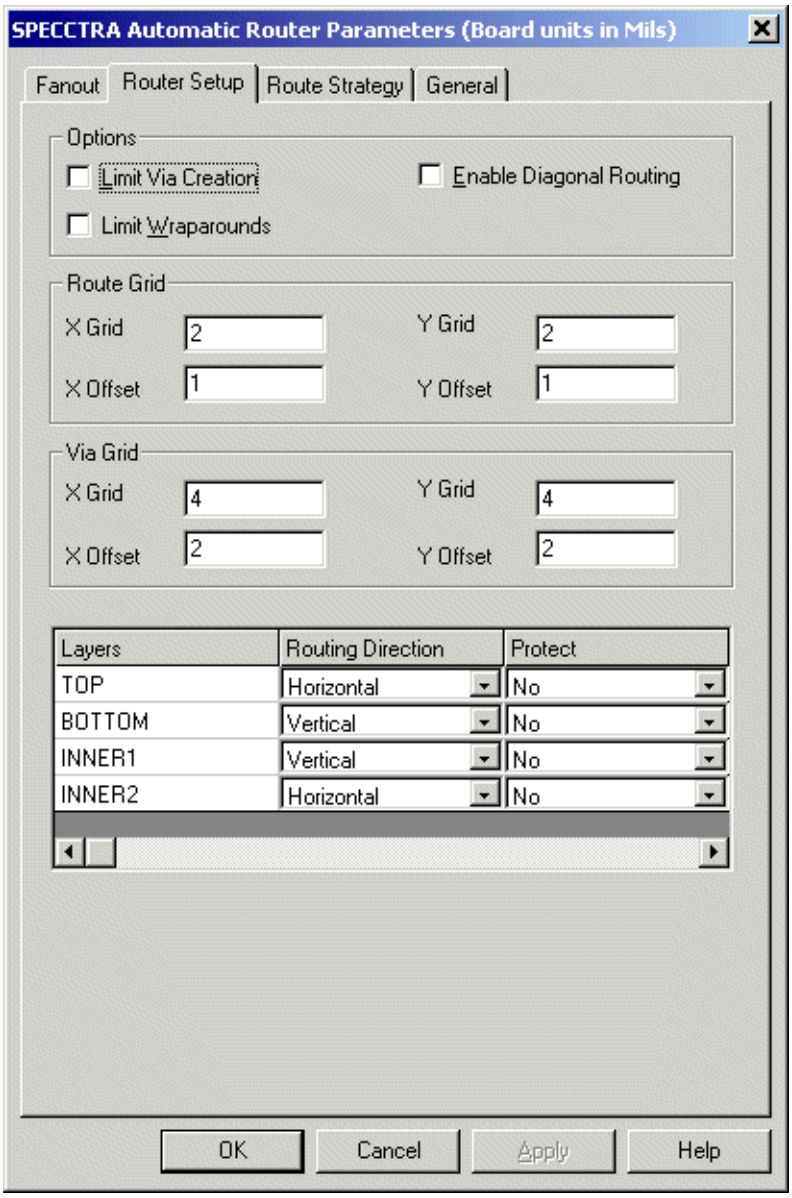
Miter after Route

Controls whether SPECCTRA does mitering after all route and clean passes are completed. No mitering is done if the routing does not converge.

Routing Passes Example

The following example describes how you can use group numbers to perform routing actions using conditional statements in the Specify Routing Passes.

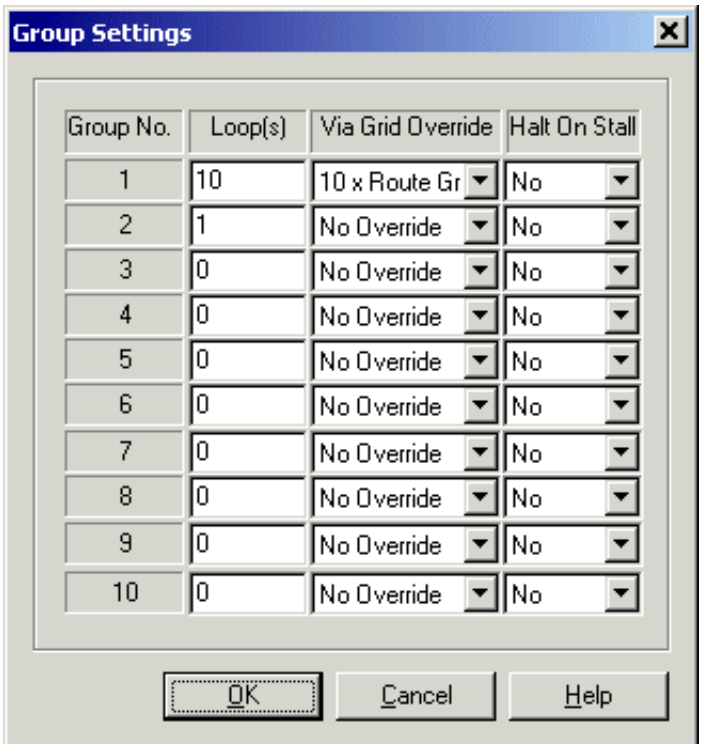
For this example, you will specify the route grid as 2 mils and the via grid as 4 mils in the Router Setup tab.



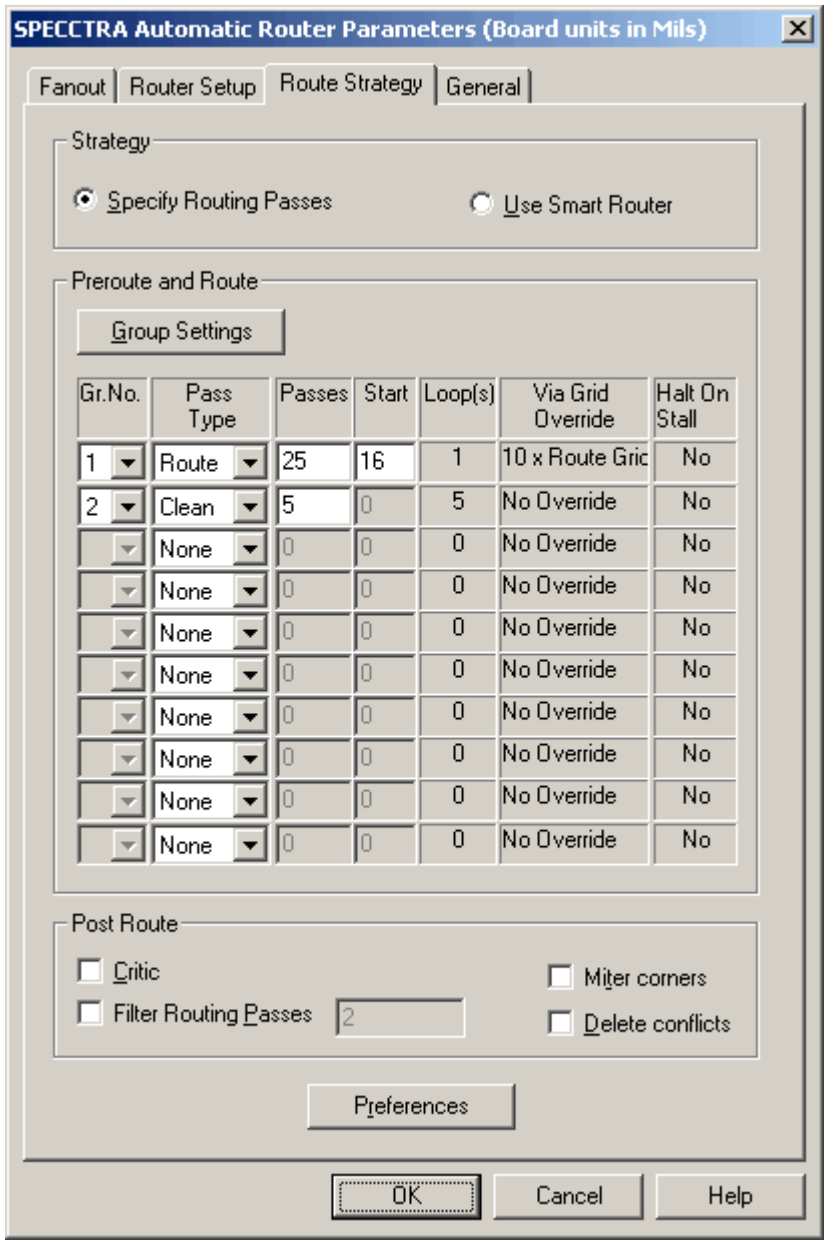
Example 1

If you want to run 25 route passes starting from the 16th route pass, in 10 loops, and using a via grid that is ten times the route grid, and then want to run five clean passes, do the following:

1 Specify the settings for group numbers 1 and 2 in the Group Settings dialog box as shown below:



2 Specify the route pass settings in the Route Strategy dialog box as shown below:



SPECCTRA will first run 25 route passes starting from the 16th route pass, in 10 loops, using a via grid of 20 mils (ten times the route grid of 2 mils specified in the Router Setup tab), and then run five clean passes.

Router Setup tab (SPECCTRA Automatic Router Parameters)

Options

Limit Via Creation	Lets you route on the active layer only and avoids creating vias on other layers.
Limit Wraparounds	Lets you route by avoiding (wherever possible) a wire that routes around a pin to get to another pin.
Enable Diagonal Routing	Lets the route and clean passes to use diagonals on all layers that you choose for routing.

Route Grid

Lets you set the X, Y route grid spacing and the offset from where the grid originates. Values are in user-defined units.

Via Grid

Lets you set the X, Y via grid spacing and the offset from where the grid originates. Values are in user-defined units.

Layers

Controls the layers to be used for routing.

Routing Direction

Lets you specify the routing direction for each routing layer.

Click on the Routing Direction drop-down list next to a layer name to select the routing direction for the layer as Horizontal or Vertical.

Protect

Click on the Protect drop-down list next to a layer name to select the protection settings for the layer. Select Yes if you want all clines on the specified layer to be fixed so they cannot be ripped up during routing.

Save Footprint As dialog box

The Save Footprint As dialog box displays when you choose the Save As button in the library manager. You can use the dialog box to save a new footprint to the library of your choice, and to copy existing footprints from one library to another (by saving a copy of the footprint to a selected library).

You can also use the dialog box to create a custom footprint library. To do so, choose the Create footprint library button, enter a name for the library, and select a target directory.

Name of footprint

If you are saving a new footprint, the name of the footprint displays in this text box. If you are saving an existing footprint (to an existing library or a new custom library), the current full path and footprint name appear in this text box.

Name of library

From this drop-down list, you can access any of the libraries that are currently available to Layout (listed in the Libraries window in the library manager). Or, you can choose the Browse button to locate and select a library.

Create New Library

Choose this button to name and select a target directory for a new, custom library. The footprint that currently displays in the Name of footprint text box is saved to the new library.

Save padstack - Select library dialog box

This dialog box appears when you choose the Save to Library command from either the pop-up menu or the Padstacks submenu (on the Tool menu). Note that, in either case, the Padstacks spreadsheet must be open and a padstack(s) must be selected.

You can use this dialog box to save padstacks to a new padstack library or an existing padstack library. If you try to save a padstack to a library and the padstack already exists in this library, Layout displays a dialog box asking if you want to replace the padstack. If you choose the No button, the attempt to save the padstack is cancelled.

Related topics

Edit Padstack dialog box

Edit Padstack Layer dialog box

Search Footprint dialog box

The Search Footprint dialog box appears when you do one of the following:

- Click the Search button in the library manager.
- Double-click the row for a part for which no footprint was found in the Configure Design Library dialog box.

  Note: If no footprint is found for a part, the text “-- Not Found --” in red color is displayed in the Footprint column next to the part name in the Configure Design Library dialog box.

You can use this dialog box to quickly search footprints by name, minimum pin count, maximum pin count or a combination of these search options.

Search by

Name

Select this check box if you want to search footprints by name.

Enter the name of the footprint you want to search for. You can use the * (asterisk) character to perform a wildcard search.

Match whole word only

Select this check box if you want to find only footprints that match the full footprint name you specified.

For example, if you enter the footprint name as VRES1 and do not select this check box, footprints with names such as VRES1, VRES10, VRES11, and so on will be found. If you select this check box, the search will only find the VRES1 footprint.

Min. pin count

Select this check box if you want to find footprints that have at least the specified number of pins and also have the same pin names as the pin names on the part in your Capture schematic.

Enter the minimum pin count.

Max. pin count

Select this check box if you want to find footprints that do not have more than the specified number of pins and also have the same pin names as the pin names on the part in your Capture schematic.

Enter the maximum pin count.

Search in

Configured libraries

Select this check box if you want to search the libraries you have made available for use in Layout. For more information on making a library available for use in Layout, see Making libraries available for use on page 531.

Other libraries

Select this check box if you want to search the libraries you have not made available for use in Layout.

Click the browse button to select the libraries.

Note: You cannot select libraries existing in multiple directories. Copy all your custom libraries to a single directory.

Search

Click this button to search for footprints that meet the specified search criteria.

Search Results

The footprints that match the search criteria are displayed in the Footprints list. The libraries in which the footprints exist are displayed in the Libraries list.

Libraries

Select a library to display the footprints in the library that match the search criteria.

Footprints

Displays the footprints that match the search criteria.

Add

Select a footprint in the Footprints list and click this button to view the footprint in the footprint editor.

Note the following:

- If the library in which the selected footprint exists is not already made available for use in Layout, the library is automatically added to the list of libraries available for use in Layout. For more information on making a library available for use, see Making libraries available for use on page 531.
- If the library in which the selected footprint exists is already made available for use in Layout, Layout displays an error message that the library is already bound to the session. Click OK to view the footprint in the footprint editor.

Related topics

Searching Footprints

Creating design libraries

Select Footprint dialog box

The Select Footprint dialog box (accessed from the Footprint button in the Add Component dialog box or the Edit Component dialog box) sets certain component properties. The Select Footprint dialog box also shows up as the Footprint for *X* dialog box, when AutoECO cannot match a component to a footprint in your design (where *X* is the name of the component in question).

The physical description of a footprint consists of three types of graphic objects, which are padstacks, obstacles (which include assembly drawing data, keepouts, silkscreens), and text. You can view the graphic representation in the library manager.

Libraries

- **Library List** - Lists the libraries that are available for the current Layout session.
- **Add** - Locate libraries and add them to the list of available libraries for the current Layout session. This list displays in the Libraries window.
- **Search** - Search footprints by name, minimum pin count, maximum pin count or a combination of these search options, using the Search Footprint dialog box.
- **Remove** - Remove selected libraries from the Libraries window.

Footprints

Lists the footprints that are contained in the libraries that are selected in the Libraries window. Select a footprint to display a graphical preview.

Related topics

Create New Footprint dialog box

Edit Footprint dialog box

Footprints spreadsheet

Select Footprint dialog box (for Design Library)

The Select Footprint dialog box appears when multiple footprints are found for parts in the Configure Design Library dialog box.

You can use this dialog box to select the required footprints for the parts.

Symbol/Capture Footprint

Displays the names of parts in your OrCAD Capture schematic for which multiple footprints were found.

Footprint

Displays the footprints found for each part. Click the drop-down list next to each part name to select the required footprint.

Related topics

Creating design libraries

Select Layer dialog box

This dialog box appears when you choose Select Layer from the View menu or when you choose Layer, Select from the Tool menu.

Layer

The layer you select from this drop-down list appears at the fore of your design window.

Related topics

Select Layer command

Select Next dialog box

Once you define a group of components using the Component Selection Criteria dialog box, and you choose Component, and then the Place command (Component) from the Tool menu, the Select Next dialog box appears.

The Select Next dialog box has either a single text box or several component options, depending upon the number of components selected with the Component Selection Criteria dialog box.

- In the first case, enter the component reference designator in the text box and choose OK. The component you specify is then attached to the pointer's crosshairs and ready to be placed.
- In the second case, simply select the component from those listed in the dialog box and choose OK. The component appears beneath the pointer crosshairs and is ready for placement.

Related topics

Component Selection Criteria dialog box

Components spreadsheet

Place command (Component)

Select Padstacks dialog box

The Select Padstacks dialog box enables you to select one or more padstack layers, or a complete padstack, based on the name and physical description of the padstack, and the individual pads on each layer of the padstack. For example, if you wanted to select all of the drill layers for padstacks that have a 37 mil drill, you could easily do so using this dialog box.

Each selected option in the dialog narrows the selection (acting as an AND operator), with the exception of the Layer list, which allows you to select pads on each of the selected layers (acting as an OR operator). If no particular layer is selected, the dialog box selects from all layers.

If you leave a check box in its initial default selection mode, it will be ignored. If you check an option, then the selected padstacks must match that selection. If you uncheck an option, then the padstacks must not match that selection.

The first five options apply to the entire padstack:

- Padstack Name

Note: This option enables you to select a padstack by name. You can use an asterisk (*) as a wildcard for any number of characters, or a question mark (?) for one character.

- Non-Plated
- Use For Test Point
- Large Thermal Relief
- Flood Pours/Planes

The remaining options apply to individual layers of the padstack:

- **Layer** - This option enables you to choose the layer or layers to which the remainder of the selections will apply. If you make a selection here, then only the relevant layers will be selected when you press OK. If you make no selection here, then the remainder of the selection criteria apply to *any* layer. Note that leaving this blank does not mean that the remaining selections must apply to all of the layers of the padstack.
- No Connection
- Pad Width
- Pad Height
- X Offset
- Y Offset

Select Padstack dialog box (Pad Array Generator)

The Select Padstack dialog appears when you press the Select button in the Pad Array Generator dialog box This dialog allows you to choose the padstacks that will be used in your array.

Padstacks box

This box lists all of the padstacks that are currently available in the configured padstack libraries. The currently selected padstack is highlighted and displayed in the Padstack Preview area. Only the default padstacks are available until you add new libraries in the Padstack Libraries box.

Note: Not all padstacks are supported in the Pad Array Generator.

Supported padstacks	Unsupported padstacks
Round	Undefined
Oblong	Oval
Square	Thermal
Rectangle	Annular
	Pads that don't have at least a top or bottom pad.

Padstack Preview area

The Padstack Preview area displays a preview of the padstack currently selected in the Padstacks box. The preview displays the top layer of the pad and, if applicable, any drill. If a top pad does not exist, the preview displays the bottom pad.

Top Pad area

In the Top Pad area, the padstack height, width, drill size and rotation are displayed.

Padstack Libraries box

The Padstack Libraries box displays the libraries that are currently in use. You can only select padstacks from libraries in this list.

To add new padstack libraries

1 Click the Add button.

2 Navigate to the library file in the Open dialog, and click OK.

To remove a library

1 Highlight the library in the Padstack Libraries box and click the Remove button.

OK

Accepts the selected padstack and returns to the Pad Array Generator dialog box.

Cancel

Closes the dialog and does not save any padstack changes.

Related topics

Pad Array Generator dialog box

SPECCTRA Reports dialog box

The SPECCTRA Reports dialog box lets you generate placement and routing reports using SPECCTRA from Layout.

The SPECCTRA Reports dialog box appears when you choose Autoroute SPECCTRA from the Auto menu, and then choose View Report.

Reports Type

Displays the list of placement and routing reports you can generate using SPECCTRA. You can generate the following reports using SPECCTRA.

Report Name	Report Filename	Description
Unconnected Nets	UNCONN.RPT	Lists all unconnected nets.
Route Status	STATUS.RPT	Lists a summary of routing data for the design. For more information, see *Routing Status Report* in the *Allegro PCB Router Command Reference*.
Conflicts	CONFLICT.RPT	Lists all components involved in placement rule violations or in placement and routing rule violations.
Corner	CORNERS.RPT	Summarizes the status of all routed corners in the design, listing corners that are 90 or 135 degree angles, arcs, and other angles. It identifies how many 90 degree corners remain after running recorner or miter commands.
Keepout	KEEPOUTS.RPT	Lists all defined keepouts (except component height and component group keepouts) and includes type, shape, layer, and coordinate information for each keepout. For more information, see *Keepouts Report* in the *Allegro PCB Router Command Reference*.
Pins without fanout vias	NOFANOUT.RPT	Lists all component pins that lack an escape wire and via after the last fanout command runs. You can use this report to determine whether pins failed the fanout operation. You can further determine whether pins are blocked or cannot escape due to rule settings.
Component Placement	PLCMENT.RPT	Lists all component locations and summarizes connection data. The report is organized by reference designator.

Report Name	Report Filename	Description
Component Property	PROPERTY.RPT	Lists properties on components. The report is organized by component.
Network	NETWORK.RPT	Lists net names, number of pins, vias, wires, tjunctions in each net, and Manhattan versus routed lengths data for each net (including one-pin nets). For more information, see *Network Report* in the *Allegro PCB Router Command Reference*.
Padstacks	PADSTACK.RPT	Lists the via, pin, and SMD padstacks from the library section of the design file.

Add

Select the report you want to generate from the Reports Type list and click the Add button to add the selected report to the list on the right hand side. The list on the right hand side displays the list of reports to be generated.

You can select multiple reports and add them at the same time.

Remove

If you do not want to generate a report, select it in the list on the right hand side and click the Remove button to remove the the report from the list of reports to be generated.

You can select multiple reports and remove them at the same time.

Status of last SPECCTRA operation

Select this check box if you want to view a report displaying the status of the last operation SPECCTRA performed on your board.

For example, after running a fanout operation using SPECCTRA, if you select this check box and click OK,

SPECCTRA displays a report named MONITOR.STS displaying the status of the fanout operation. The MONITOR.STS report is saved in the project directory.

Use a Customized DO File

Select this check box if you want to use a customized .DO file for generating reports using SPECCTRA.

Enter the name and path to the .DO file, or click the Browse button to select the file.

Edit

Click Edit if you want to edit the specified .DO file.

OK

Click this button to generate the reports.

The report files are created in the project directory and displayed in a text editor.

Related topics

Generating SPECCTRA Reports

SPECCTRA to Layout dialog box

In the session frame, from the File menu, point to Export and choose SPECCTRA to Layout to invoke this dialog box.

Input SPECCTRA File

In this text box, enter the path and filename of the SPECCTRA file that you want to translate into Layout.

Output Layout File

In this text box, enter the path and filename of the Layout board file that you want to create.

Overwrite existing files

By selecting this option, you dispense with any warnings that Layout normally provides before overwriting files.

Original Layout File

Related topics

Layout to SPECCTRA dialog box

Layout to SPECCTRA command (Export)

SPECCTRA to Layout command (Import)

Stackup Editor dialog box

The Stackup Editor dialog box maps a logical stackup of OrCAD Layout to a physical stackup on a PCB. Using this dialog box, you can also define some of the physical properties of a PCB, such as the type of dielectric to be used, whether solderpaste, solder mask and silkscreen should be used, and display them as a non-electrical component on one of the Layout documentation layers.

Technology

Specifies the technology used during the PCB manufacture process. Stackup Editor support two technologies, Core and Builtup.

When you select the Core technology, the dielectric material used between two layers of the PCB will alternate between Core and Prepeg.

If the manufacturing technology selected is builtup, the dielectric material used between two layers of the PCB is Prepeg.

Solder Paste

Select this check box if you want a solder paste to be used on the top or bottom layer of the printed circuit board. To specify the properties of the solder paste, click the enabled browse button.

Solder Mask

Select this check box if you want a solder mask to be applied on the top or the bottom layer.

In the properties dialog box, specify the type of material and also the dielectric constant for the material that is to be used for masking.

SilkScreen

Select this check box to enable printing on silkscreen on the top layer of a PCB.

When you select this option, OrCAD Layout checks if the silkscreen layer is enabled in the post processing settings. If not you may receive a message stating if the silk screen layer in Post Process Settings is to be enabled.

Foil Layers

Select the appropriate check box to indicate the layer over which you want the apply the foil layer. A foil is a coated copper layer on the board. To know how to specify properties of a foil, see Plating Properties dialog box.

When you add a foil layer, the physical definition of your stackup might change to ensure that the outermost layers are prepreg layer.

Advanced Options

Select this to specify overall properties of the board. To know more about the advanced options, see Advanced Options dialog box.

Drawing Layer

This field is available only if you launch the Stackup Editor from Layout.

Specifies the documentation layer on which board stackup is to be placed. The stackup drawing is inserted as a non-electrical component, that can be placed any where on the doc layer. The stackup drawing is routing disabled and does not appear in the backannotation data.

Via protection

This field is available only if you launch the Stackup Editor from Layout. Select the via protection method that should be used while manufacturing the printed circuit board.

Cross Section View

Displays the cross-sectional view of the printed circuit board. If required, you can change the order in which various PCB layers appear in the stack, using the button provided within the view.

Click this button to insert a routing or a signal layer to the stackup.

Click this button to insert a power plane to the stackup.

Click this button to remove the selected layer from stackup.

Note: You cannot delete a layer with routing information.

Click this button to move the selected layer up in the stackup.

Note: You cannot move the TOP and BOTTOM layers, only inner layers can be moved up and down.

Click this button to move the selected layer down in the stackup.

Click this button when you want to display the Properties dialog box for the selected layer.

To know more about the Properties dialog box, see Properties (Layer) dialog box.

Sweep Edit dialog box

To invoke the Sweep Edit dialog box, select the Spreadsheet tool, choose the Strategy command, choose the Route Sweep command, then double-click on the sweep or the cell you want to edit.

The Route Sweep option you use determines the manner in which Layout routes your board when you invoke the Board command (Autoroute).

Layout divides your board into a number of equally-sized areas, comprising rows and columns. The number of areas into which Layout divides the board is a function of size of the DRC box and the board grid. When you invoke the Batch Route command, Layout places routes within the area bounded by the DRC box. Once it has completed this routing, it moves the DRC box up or down (depending on the direction specified in the Sweep Edit dialog box) and routes that area of the board. It then completes routing for the column in which it was invoked, before moving to another column of the board. Again, the direction in which Layout moves the DRC box is determined by the specifications in the Sweep Edit dialog box.

Sweep Name

The default Sweep Name reflects its normal usage in the routing of the board. The name of the current sweep is displayed in the title bar at the top of the screen during routing, so that you can tell at a glance which stage of the routing is in progress. If more than one sweep is selected, this specifies the number of selected sweeps.

Diagonal Routing

Off	This will prohibit the router from installing any 45 degree angles during routing.
On	This will enable 45 degree routing, but only for memory heuristics, and where necessary to clear pads and obstacles during routing. Use the On option for routing VCC and ground.
Maximize	When you enable Maximize, the router will use 45 degree routing wherever possible, unless a 90 degree angle is necessary to clear pads and obstacles during routing. For best manufacturing, it is recommended that you always route your boards using 45 degree angles. By using the 45 degree routing, you will also minimize the number of segments in the design.

Sweep Direction

Layout offers you a choice of the following directional specifications for the route sweeps:

- **Down, Left** - When you specify Down, Left for the Route Sweep Dialog, Layout routes the design in vertical swaths. It begins with the area in which the autorouter was invoked, completes that swath to the bottom edge of the board, then moves to the upper edge to finish routing that swath. It then follows the same pattern in the vertical swath that is to the left of the original, and finishes by following this pattern in the vertical swath at the right side of the board.

- **Down, Right** - When you specify Down, Right for the Route Sweep Dialog, Layout routes the design in vertical swaths. It begins with the area in which the autorouter was invoked, completes that swath to the bottom edge of the board, then moves to the upper edge to finish routing that swath. It then follows the same pattern in the vertical swath that is to the right of the original, and finishes by following this pattern in the vertical swath at the left side of the board.
- **Left, Down** - When you specify Left, Down for the Route Sweep Dialog, Layout routes the design in horizontal swaths. It begins with the area in which the autorouter was invoked, completes that swath to the left edge of the board, then moves to the right edge to finish routing that swath. It then follows the same pattern in the horizontal swath that is below the original, and finishes by following this pattern in the horizontal swath at the upper edge of the board.
- **Left, Up** - When you specify Left, Up for the Route Sweep Dialog, Layout routes the design in horizontal swaths. It begins with the area in which the autorouter was invoked, completes that swath to the left edge of the board, then moves to the right edge to finish routing that swath. It then follows the same pattern in the horizontal swath that is above the original, and finishes by following this pattern in the horizontal swath at the lower edge of the board.
- **Right, Down** - When you specify Right, Down for the Route Sweep Dialog, Layout routes the design in horizontal swaths. It begins with the area in which the autorouter was invoked, completes that swath to the right edge of the board, then moves to the left edge to finish routing that swath. It then follows the same pattern in the horizontal swath that is below the original, and finishes by following this pattern in the horizontal swath at the upper edge of the board.
- **Right, Up** - When you specify Right, Up for the Route Sweep Dialog, Layout routes the design in horizontal swaths. It begins with the area in which the autorouter was invoked, completes that swath to the right edge of the board, then moves to the left edge to finish routing that

swath. It then follows the same pattern in the horizontal swath that is above the original, and finishes by following this pattern in the horizontal swath at the lower edge of the board.

- **Up, Left** - When you specify Up, Left for the Route Sweep Dialog, Layout routes the design in vertical swaths. It begins with the area in which the autorouter was invoked, completes that swath to the top edge of the board, then moves to the lower edge to finish routing that swath. It then follows the same pattern in the vertical swath that is to the left of the original, and finishes by following this pattern in the vertical swath at the right side of the board.
- **Up, Right** - When you specify Up, Right for the Route Sweep Dialog, Layout routes the design in vertical swaths. It begins with the area in which the autorouter was invoked, completes that swath to the top edge of the board, then moves to the lower edge to finish routing that swath. It then follows the same pattern in the vertical swath that is to the right of the original, and finishes by following this pattern in the vertical swath at the left side of the board.
- **Route Next Connection** - The default Next sweeps immediately follow the Maze Route sweep. If you find that the board completion is less than 95% after two Next sweeps, you may need to look at the placement or the basic design criteria of the board. For example, you could change the grid from 25 to 8 1/3 (track widths should go from 12 to 8, and spacing should go from 12 to 7) to finish the board.

Note: For the best results, use the density graph to determine the area of the design with the greatest connection density. The autorouter will have greater success when invoked from the most dense area of the design.

Routing box X, Y

With this option, you specify the size of the routing window. The most reliable way to route a board is to make the routing window as large as your computer's RAM will permit:

- For boards with 25 and 20 mil grids: the default routing window size is 250 x 200 and the maximum is 350 x 300.
- For boards with 12 1/2, 10, 8 1/3 mil grids: the minimum default window size is 250 x 200 and the maximum is 450 x 350.

If you have 16 MB of physical RAM, it is recommended that you use no more than 12 MB when starting a route; if you have 32 MB physical RAM, you should use a maximum of 24 MB when starting a route.

The requirement for RAM increases as the routing window size increases and as the number of enabled routing layers increases. In addition, the time required to complete the routing operation increases as the size of the routing window increases.

If you have too little RAM for the routing, you'll notice a lot of disk swapping activity.

If your board dimensions allow, create a routing window that spans the board. This allows the router to optimize the use of routing channels that run the length of the board.

Overlap X, Y

The Overlap percent determines the amount of space the router uses to reroute tracks as it moves from window to window. It is recommended that you leave these values at the default 26%.

If the overlap is too small, you could end up with many small partial routes. If the overlap is too big, the router will tend to try to rip up unneeded routes.

Related topics

Load Strategy command (File menu)

Save As Strategy command

System Settings dialog box

Layout has five distinct grid settings and five distinct grids; they are all available in the System Settings dialog box. In addition, the System Settings dialog box lets you set rotation angles and increments, as well as workspace settings. To display the System Settings dialog box, choose System Settings from the Options menu.

Display Units

Select the unit of measure you wish to use for the five grids on the right side of the dialog box. The available options are: mils (m), inches (in), microns (u), millimeters (mm) and centimeters (cm).

Display Resolution

The integer in this text box determines the gradation of the Cartesian coordinates Layout uses to indicate the position of the pointer. There are two places Layout displays these coordinates: the lower, left-hand corner of the application's window and directly below the toolbar.

Grids

Layout uses five discrete grids to determine the position of various electronic entities; separate grids are used for vias, routing and component placement. Additionally, there is a setting for the visible grid (that which appears over the design window) and a setting for "details" (obstacles and text objects).

Rotation

These settings determine the behavior of the Rotate command.

Increment

The number of degrees a component or object will rotate when you choose the Rotate command.

Snap

Assigns the very finest rotation increment for use when the Place grid value and the Increment value are very small. This is especially useful on a round board for precise angular placement of components.

For example, if you specify the rotation increment angle as 10 degrees and the snap angle as 3 degrees, the component will be rotated by 9 degrees (the finest rotation increment).

Related topics

User Preferences command

Test Point Settings dialog box

A test point is a special point of access to an electrical circuit that is used for electrical testing purposes. The Test Point Settings dialog box allows you to specify certain rules for the creation, location and spacing of test points. This dialog box appears when you choose Test Point Settings from the Option menu.

Generate test points from vias

This option creates test points from existing vias.

Allow test points under components

With this setting selected, you are able to create test points under components. If you do not select this option, then test points are created outside of component outlines.

Allow through-hole as test points

When you select this option, Layout designates through-hole pins as test points.

Test point pitch

The value you enter in this text box determines the minimum space allowed between test points.

Related topics

Via Selection dialog box

Test Points command (Place)

Test Point Selection Criteria dialog box

The Test Point Selection Criteria dialog box appears as a result of choosing the Select From Spreadsheet command in the Tool, Test Points menu. Use this dialog box to locate and select test points listed in the Components spreadsheet.

You can limit or broaden your search for a test point with the three drop-down lists and Exclude Locked option. When you choose the OK button, the test points that meet your criteria are located in the Components spreadsheet and highlighted.

Padstack Name

By specifying a specific padstack or via with the Padstack Name drop-down list, you can select the test point associated with this via or padstack. Note: using the Test Point Settings dialog box, you can create test points from vias automatically.

Net Name

To highlight test points associated with a particular net, select the appropriate net name from this drop-down list.

Group Number

As with components, test points can be categorized according to groups. To find and highlight test points within a specific group, enter the relevant group number in this text box.

Exclude Locked

Just as with components, you may lock test points in place on the printed circuit board. If you want to exclude all locked test points from the selection process, enable this option.

Related topics

Via Selection dialog box

Test Points command (Place)

Text Edit dialog box

To display the Text Edit dialog box, choose the New or the Properties command from the Tool, Text menu (you can choose the latter command only while text is selected in the design window). Or, choose the text toolbar button, select existing text, then choose Properties from the pop-up menu.

Text String

This is the text string that is added to the board.

Type of text

There are a number of text types:

- **Free** - Normal text, attached to the board or to a component. Free text doesn't have properties associated with it.
- **Reference Designator** - Automatically generated from the component.

- **Component Value** - Component-attached text corresponding to the component value.
- **Custom Properties** - Properties added in Capture that are brought in as part of the component.
- **Package name** - Automatically generated. Assumes the schematic part name if no package is assigned.
- **Footprint name** - Automatically generated.

Line Width

Assigns a line width for drawing the text.

Rotation

Assigns rotation in degrees and minutes of the entire text string.

Radius

Assigns an overall radius for the text string.

Text Height

Assigns text height. This is the most common way of assigning overall text size.

Char Rot

Assigns individual character rotations.

Char Aspect

Assigns the width (relative to the height) of the text characters.

Mirrored

Mirrors the text.

Layer

Specifies a layer for the text. The Global layer is layer 0.

Comp Attachment

Displays the Comp Attachment dialog box, so that you can attach the text to a component. This option doesn't display if you select footprint text in the library manager.

Related topics

Text Editor command (File menu)

Comp Attachment dialog box

Text Selection Criteria dialog box

The Text Selection Criteria dialog box appears when you choose the Find button on the toolbar, or the Select Any command from the pop-up menu command, while the Text spreadsheet is active. You can locate a specific component, within the spreadsheet, by typing in the text string. You can also find a group of components that meet specified search criteria, by selecting the appropriate option, and text string and layer information if known.

Text String

Specifies the string of text.

Type of Text

- **Free** - Specifies that the text to search for is text attached to a board location.
- **Footprint Name** - Specifies that the text to search for is attached to the footprint, though it is not a footprint name.
- **Custom Properties** - Specifies that the text to search for is component text other than the component name or pin name.
- **Reference Designator** - Specifies that the text to search for is the footprint name, and is automatically generated.
- **Package Name** - Specifies that the text to search for is a package name, and is automatically generated. Layout uses the part name if no package has been assigned to the component.
- **Component Value** - Specifies that the text to search for is a component name, and is automatically generated.

Layer

Specifies the layer on which the text displays. Conn indicates the text displays in all layers.

Related topics

Comp Attachment dialog box

Thermal Relief Settings dialog box

In this dialog box, you specify measurements for both small thermal relief and large thermal reliefs. Specify measurements as integers using the units you have selected in the System Settings dialog box.

Small thermal reliefs are used by Layout in all cases unless you specifically choose the large thermal relief.

To display this dialog box, choose the Thermal Relief Settings command from the Options menu.

The explanations below apply to both the Small Thermal Relief and Large Thermal Relief settings groups. Click the Thermal Relief button to see a graphic presentation of a thermal relief and its properties.

{button Thermal Relief,JumpId(layout.hlp>lay_pic,Thermal_Relief_Graphic)}

Annular Over Drill

As a practical matter, the interior size of the thermal relief should be no greater than 50 mils.

Isolation Width

Though you can enter larger values, the isolation width should be no greater than 50 mils.

Spoke Width

As a practical matter, the spoke width should be no greater than 100 mils.

Related topics

Thermal Relief Settings command

System Settings command

Track Width dialog box

To display the Track Width dialog box, choose one of the routing tools, select a track, and then choose the Change Width command (Track Segment) from the Tool, Track Segment menu.

New Width

Specifies the width of a track. You can change this value while you are routing, or after the track is completely routed. If you specify a new width that is outside the MIN and MAX range shown in the Nets spreadsheet, the new value is written into the spreadsheet. The dialog box remembers the last width you created, so if you select, for example, a 5 mils track, and change its width to 12, the dialog reads width "12," even if the next track you select is 50 mils.

User Preferences dialog box

The User Preferences dialog box appears when you choose the User Preferences command from the Options menu. By choosing the Save User Preferences button, you can save the settings you select and clear in your local directory; future layout settings use these saved settings.

Display Preferences

Enable Full-screen Cursor

This setting extends the cross hairs of the pointer to the limits of the design window, allowing you to align distant parts.

Enable Auto Pan

If you select this option, Layout moves the design window so that it is centered on the location of your pointer. The design window moves only when the pointer is at the design window's edge.'

Use Opaque Graphics

If you select this option, Layout superimposes the active layer over any other visible layers (as indicated in the Colors spreadsheet).

Use Hollow Pads

If you select this option, pads appear as hollow circles or squares, thereby reducing the time required to redraw the design window.

Show 3D Effects

If selected, this option indicates a component's vertical dimension (if any) by placing a beveled grey graphic over the component's footprint. The height of the component is also specified quantitatively (using the unit of measure entered in the System Settings dialog box).

Global Preferences

Activate Online DRC

Enables online design rule checking. Equivalent to choosing the design rules check toolbar button.

Allow DRC Errors

Enables DRC errors with markers, created in real time to show where violations are occurring as you route or place a board. These errors are temporary and only apply to the object being altered.

Instantaneous Reconnection Mode

Enables reconnect mode, which you can use to show or hide nets. Equivalent to choosing the reconnect toolbar button.

Allow Editing of Footprints

This option permits you to edit component footprints on the board using the Obstacle and Pin tool without having to open the Library Manager.

Copper Pour Preferences

Enable Copper Pour

By selecting this option, you allow Layout to fill copper area and copper pour obstacles. This can be done by choosing the refresh all toolbar button (represented by a red exclamation mark on the toolbar) or by choosing Refresh, All or Refresh, Repour from the Auto menu.

Use Fast Fill Mode

Reduces the drawing time for copper pour by using a simple pattern to represent copper pour on your screen. This option only affects the display of the copper pour on the screen; it does not accelerate the actual pour process. You must disable this option before you create a Gerber plot.

Use Pours for Connectivity

Layout considers ratsnests to be completed when they exist in a copper pour, provided that the copper pour is common to the same net as the pins.

Miscellaneous Preferences

Show Tooltips

With this setting selected, Layout displays a brief explanation of each tool on the toolbar when the pointer is positioned over the tool icon.

Activate Auto Tool Select Mode

If selected, you can select any object without having to set to the object tool type first. This is only for selecting a single object, not for block selection. When this option is selected, selecting an object causes Layout to change to the

appropriate tool for the selected object. For example, if you select a component, Layout changes to the Component tool.

Move Free Vias With Components

With this option selected, Layout behaves exactly as it did in version 9.1; free via fanouts will move with components, and if you CTRL + Click a component, you will be asked if you want the free vias to move with the component. If this option is turned off, free vias, including free via fanout patterns, will not move with components.

All Traces follow a Moved Via

With this option selected, the traces attached to a via remain attached to the via when you move the via. If this option is turned off, when you move a via, the traces attached to the via get unrouted and display in Layout as rats.

Place component from Capture

With this option selected, you can cross-place components from OrCAD Capture to Layout.

When you select a component in Capture and move the cursor over the Layout window, the component is attached to the cursor in a place (or move) mode. For more information on cross-placing components from Capture to Layout, see Cross-placing components from Capture to Layout on page 333.

Note: Layout remembers this setting only for the current session. The next time you start Layout, you have to select this check box again if you want to cross-place components from Capture to Layout.

Activate Selection Filter

With this option selected, you can use the shortcut keys to select the objects from the current object selection mode that are closest to the current cursor position. This option is very

useful for selecting objects on a densely populated layout board, because you use shortcut keys to select objects as well as to switch between various object selection modes, such as component selection mode or obstacle selection mode.

Allow renumbering of pins

With this option selected, Layout automatically renumbers numeric pin names when you add or delete pins in the Footprint Editor. For example, if you have pins numbered from 1 to 8 on your footprint and you delete pin 7, pin 8 will be automatically renumbered to 7.

Clear this check box if you do not want numeric pin names to be automatically renumbered when you add or delete pins in the Footprint Editor.

Minimum Track Width to Display

With this option selected, Layout only displays tracks that are wider than the amount you specify; it draws all other tracks as a single pixel line.

Save User Preferences

Click this button to save the settings to the layout.ini file.

Related topics

Global Spacing command

Route Settings command

Colors command

Via Selection dialog box

The Via Selection dialog box appears when you choose Change Via Type command from the Tool, Via menu.

Select Via Type

Select a new via type. The number of types available to you is dependent upon the number of types defined in the Padstacks spreadsheet.

Workspace Settings dialog box

The Workspace Settings dialog box appears when you choose the Workspace Settings button located on the System Settings dialog box. Within the context of Layout, “workspace” is defined as the total potential area of your design, as it is modeled electronically.

Current Workspace

The information displayed here indicates the current size of your workspace as specified in the settings group below.

New Workspace

There are three sizes of workspace you can select; each size (Very Small, Typical, Very Large) has both a metric and a standard setting. The metric setting is graduated in millimeters and the standard is in inches.

Convert

OrCAD recommends that database conversions be performed only at the beginning of a design, before a netlist has been loaded. Conversions of the database must be synchronized with the equivalent schematic unit conversions.

Related topics

User Preferences dialog box

System Settings dialog box

Layout Session Window Commands

19

File Menu Commands (LSession) on page 849

View Menu Commands (LSession) on page 864

Tools Menu Commands (LSession) on page 865

File Menu Commands (LSession)

Import command (File menu) on page 849

Export command (File menu) on page 856

1, 2, 3, 4 file commands (File menu) on page 863

Import command (File menu)

Using the Import menu in the session frame, you can import a design from a printed circuit board application other than Layout, place and route the design using Layout, then export the board back to its printed circuit board application. For example, a board created in PADS can be imported to Layout for placing and routing, then exported to PADS.

Use this command to import files from another application. Choose a command from the menu that displays:

MIN Interchange command (Import) on page 850

MAX ASCII Netlist command (Import) on page 852

PCB II Netlist command (Import) on page 852

Futurenet Netlist command (Import) on page 852

PCB 386+ command (File menu, Import command) on page 853

CadStar PCB command (Import) on page 853

PADS PCB command (Import) on page 853

P-CAD PCB command (Import) on page 854

Protel PCB command (Import) on page 854

Protel99 SE command (Import) on page 854

Tango PCB command (Import) on page 854

DXF to Layout command (Import) on page 855

IDF to Layout command (Import) on page 855

SPECCTRA to Layout command (Import) on page 855

GenCAD to Layout command (Import) on page 855

Shortcuts

Keyboard: ALT, F, I

MIN Interchange command (Import)

With the MIN Interchange command, you can translate an OrCAD ASCII format file with a .MIN extension to an OrCAD binary file with a .MAX extension.

To use the MIN translator

1 From the File menu in the session frame, point to Import and choose MIN Interchange.

2 In the MAX Interchange Notation Input dialog box, select a file with a .MIN extension, then choose the OK button.

3 In the MAX Board Output dialog box, enter a filename with a .MAX extension, then choose the Save button.

4 When the message "Processing completed" displays, choose the OK button to dismiss the dialog box.

Note: If you encounter problems converting your design, choose MAX Interchange from the Export menu in the session frame, enter the filename of your newly created .MAX file in the MAX Board Input dialog box that displays, and choose the OK button. When the MAX Interchange Notation Output dialog box displays, enter a filename with a .MIN extension, then choose the OK button. When the Options dialog box displays, accept the default of "All of the above" by choosing the OK button. When the MIN Back Annotator dialog box displays with the message "Processing completed," choose the OK button to dismiss the dialog box. At this point, you can run steps 1 through 4 in the procedure above again, in order to translate your new .MIN file to a new .MAX file.

Note: You can merge a .MIN file and a .MAX file to form a new .MAX file by closing OrCAD Layout for Windows and using the command line:

MAXMINB.EXE [inputMINfile] [outputMAXfile] [mergeMAXfile]

Note: You can dump the contents of a Layout MAX file to a .MIN file in ASCII format by closing Layout and using the command line:

MAXMINW [inputMAXfile] [outputMINfile]

This is especially useful when your design file exhibits problems that do not respond to corrective measures. You can often export the file in ASCII format and import it again as a MAX, and find that the problems are gone.

MAX ASCII Netlist command (Import)

You can use an ASCII file to edit your board, or create a design from a schematic capture application that supports ASCII output. After you create an ASCII file, you can edit the file in any text editor before reading it into Layout.

This command is most often used to create a parts list manually.

If you have a schematic capture application that creates an ASCII file in the Layout format, you can import the file into Layout to create a new design. You can also create the ASCII netlist using a text editor.

Shortcut

Keyboard: ALT, F, I, A

PCB II Netlist command (Import)

OrCAD PCB II netlists are ASCII formatted netlists that can be brought into Layout.

Layout netlists are typically binary files with an MNL extension. Layout netlists are usually created by using the Layout tab in Capture. This causes DSN2MNL.DLL to run, creating an MNL file. You can also create an ASCII netlist file in FutureNet or MAX ASCII format.

Shortcut

Keyboard: ALT, F, I, I

Futurenet Netlist command (Import)

FutureNet netlists are ASCII formatted netlists that can be brought into Layout.

Layout netlists are typically binary files with an MNL extension. Layout netlists are usually created by using the Layout tab in Capture. This causes DSN2MNL.DLL to run,

creating an MNL file. You can also create an ASCII netlist file in OrCAD PCB II or MAX ASCII format.

Shortcut

Keyboard: ALT, F, I, F

PCB 386+ command (File menu, Import command)

You can move a PCB 386+ design into Layout.

If translation problems arise, Layout displays error messages and writes them to a file in the output directory using the name of your output file and .ERR as an extension.

Shortcut

Keyboard: ALT, F, I, 3

CadStar PCB command (Import)

This translator reads CDI ASCII files from CadStar through version 7 and translates them to Layout MAX (binary) format. You can also output from version 8 if you select the (version 7) output option.

Shortcut

Keyboard: ALT, F, I, C

PADS PCB command (Import)

This translator reads ASCII files from PADS-PCB and PowerPCB and translates them to Layout MAX (binary) format.

Shortcut

Keyboard: ALT, F, I, D

P-CAD PCB command (Import)

This translator reads "PDIF" ASCII files from P-CAD Master Designer versions 3.0 through 8.5 (the final release of Master Designer) and translates them to Layout MAX (binary) format.

Shortcut

Keyboard: ALT, F, I, P

Protel PCB command (Import)

This translator reads PCB ASCII files from Protel through version 2.8 and translates them to Layout MAX (binary) format. You can also output from version 3 and later using the 2.8 ASCII output option.

Shortcut

Keyboard: ALT, F, I, L

Protel99 SE command (Import)

This translator reads PCB ASCII files from Protel 99SE and translates them to Layout MAX (binary) format.

Shortcut

Keyboard: ALT, F, I, L

Tango PCB command (Import)

This translator reads PCB ASCII files from Tango (all DOS versions) and translates them to Layout MAX (binary) format.

Shortcut

Keyboard: ALT, F, I, T

DXF to Layout command (Import)

DXF import and export are ideally suited to translating a Layout board file to AutoCAD, adding mechanical or documentation drawings and text, then returning to Layout.

The MAXDXF.EXE file translates a drawing exchange format (.DXF) file and produces an OrCAD binary file with a .MAX extension, or translates a .MAX file to the .DXF format.

Note: Translation between .MAX and .DXF files requires that the file MAXDXF.INI be in the LAYOUT directory.

Note: If you are merging multiple DXF files with data in your .MAX file, do not select the Remove existing obstacles and text option in the DXF Import dialog box. If you want to replace the board outline and its annotations with entities from the .DXF file, select the Remove existing obstacles and text option.

Shortcut

Keyboard: ALT, F, I, X

IDF to Layout command (Import)

Use this command to translate Intermediate Drawing Format (IDF) files into Layout.

IDF to Layout dialog box

SPECCTRA to Layout command (Import)

Use this command to translate SPECCTRA files into Layout.

SPECCTRA to Layout dialog box

GenCAD to Layout command (Import)

Use this command to translate GenCAD files into Layout.

GenCAD to Layout dialog box

Export command (File menu)

Using the Export menu in the session frame, you can export the board back to its printed circuit board application. For example, a board created in PADS can be imported to Layout for placing and routing, then exported to PADS.

You cannot create a board in Layout, then try to export it to another application---the board must be created in an application other than Layout if you want to export it. Also, you cannot import a board created in one application, such as PADS, place and route it using Layout, then export it to an application other than PADS---the import application must be the same as the export application.

When you export a design, only the placement and routing information (such as placement side, X and Y locations, and component rotation) is included in the export file.

Choose a command from the menu that displays:

MIN Interchange command (Export) on page 857

MAX ASCII Netlist command (Export) on page 858

PCB 386+ command (Export) on page 859

CadStar PCB command (Export) on page 859

PADS PCB command (Export) on page 859

P-CAD PCB command (Export) on page 860

Protel PCB command (Export) on page 860

Tango PCB command (Export) on page 861

Layout to DXF command (Export) on page 861

Layout to IDF command (Export) on page 862

Layout to GenCAD command (Export) on page 862

Layout to IPC-356 command (Export) on page 862

Layout to Hyperlynx command (Export) on page 862

Layout to SPECCTRA command (Export) on page 863

Layout to GenCAM command (Export) on page 863

Layout to ODB++ command (Export) on page 863

Shortcuts

Keyboard: ALT, F, E

MIN Interchange command (Export)

With the MIN Interchange command, you can translate an OrCAD binary file with a .MAX extension to an OrCAD ASCII format file with a .MIN extension. You can include as much or little information in the MIN file as desired.

To translate a MAX file into an OrCAD MIN ASCII file

1. From the File menu in the session frame, point to Export and choose MIN Interchange.
2. In the MAX Board Input dialog box, select a file with a .MAX extension, then click OK.
3. In the MAX Interchange Notation Output dialog box, enter a filename with a .MIN extension, then click Save.
4. In the Options dialog box, check the options for each data type to include. To include all data types, check the All of the above option. Click OK to continue.
5. When the message "Processing completed" displays, choose the OK button to dismiss the dialog box.

Note: If you encounter problems converting your design, choose MAX Interchange from the Export menu in the session frame, enter the filename of your newly created .MAX file in the MAX Board Input dialog box that displays, and choose the OK button. When the MAX Interchange Notation Output dialog box displays, enter a filename with a .MIN extension, then choose the OK button. When the Options dialog box displays, accept the default of "All of the above" by choosing the OK

button. When the MIN Back Annotator dialog box displays with the message "Processing completed," choose the OK button to dismiss the dialog box. At this point, you can run steps 1 through 4 in the procedure above again, in order to translate your new .MIN file to a new .MAX file.

Note: You can merge a .MIN file and a .MAX file to form a new .MAX file by closing OrCAD Layout for Windows and using the command line:

MAXMINB.EXE [inputMINfile] [outputMAXfile] [mergeMAXfile]

Note: You can dump the contents of a Layout MAX file to a .MIN file in ASCII format by closing Layout and using the command line:

MAXMINW [inputMAXfile] [outputMINfile]

This is especially useful when your design file exhibits problems that do not respond to corrective measures. You can often export the file in ASCII format and import it again as a MAX, and find that the problems are gone.

Shortcut

Keyboard: ALT, F, E, M

MAX ASCII Netlist command (Export)

You can use an ASCII file to edit your board, or create a design from a schematic capture application that supports ASCII output. After you create an ASCII file, you can edit the file in any text editor before reading it into Layout.

This command is most often used to create a parts list manually.

If you have a schematic capture application that creates an ASCII file in the Layout format, you can import the file into Layout to create a new design. You can also create the ASCII netlist using a text editor.

Shortcut

Keyboard: ALT, F, E, A

PCB 386+ command (Export)

You can move a PCB 386+ design into Layout.

If translation problems arise, Layout displays error messages and writes them to a file in the output directory using the name of your output file and .ERR as an extension.

Shortcut

Keyboard: ALT, F, E, 3

CadStar PCB command (Export)

This translator reads Layout MAX (binary) files which have previously been translated in from CadStar, and translates them back to "CDI" ASCII format CadStar files. Note that you must have the original CDI file from which the Layout MAX file was created in order to complete this translation, because the routing and placement information from the MAX file is merged in with the original database during translation.

This translator translates only routing and placement information back to the original system. Changes to padstacks, obstacles, text, and other miscellaneous data objects will not be back annotated, but will be retained in their original form.

Shortcut

Keyboard: ALT, F, E, C

PADS PCB command (Export)

This translator reads Layout MAX (binary) files which have previously been translated in from PADS, and translates them back to "asc" ASCII format PADS files. Note that you must

have the original asc file from which the Layout MAX file was created in order to complete this translation, because the routing and placement information from the MAX file is merged in with the original database during translation.

This translator translates only routing and placement information back to the original system. Changes to padstacks, obstacles, text, and other miscellaneous data objects will not be back annotated, but will be retained in their original form.

Shortcut

Keyboard: ALT, F, E, D

P-CAD PCB command (Export)

This translator reads Layout MAX (binary) files which have previously been translated in from P-CAD, and translates them back to "PDF" ASCII format P-CAD files. Note that you must have the original PDF file from which the Layout MAX file was created in order to complete this translation, because the routing and placement information from the MAX file is merged in with the original database during translation.

This translator translates only routing and placement information back to the original system. Changes to padstacks, obstacles, text, and other miscellaneous data objects will not be back annotated, but will be retained in their original form.

Shortcut

Keyboard: ALT, F, E, P

Protel PCB command (Export)

This translator reads Layout MAX (binary) files which have previously been translated in from Protel, and translates them back to "PCB" ASCII format Protel files. Note that you must have the original PCB file from which the Layout MAX file was

created in order to complete this translation, because the routing and placement information from the MAX file is merged in with the original database during translation.

This translator translates only routing and placement information back to the original system. Changes to padstacks, obstacles, text, and other miscellaneous data objects will not be back annotated, but will be retained in their original form.

Shortcut

Keyboard: ALT, F, E, L

Tango PCB command (Export)

This translator reads Layout MAX (binary) files which have previously been translated in from Tango, and translates them back to "PCB" ASCII format Tango files. Note that you must have the original PCB file from which the Layout MAX file was created in order to complete this translation, because the routing and placement information from the MAX file is merged in with the original database during translation.

This translator translates only routing and placement information back to the original system. Changes to padstacks, obstacles, text, and other miscellaneous data objects will not be back annotated, but will be retained in their original form.

Shortcut

Keyboard: ALT, F, E, T

Layout to DXF command (Export)

DXF import and export are ideally suited to translating a Layout board file to AutoCAD, adding mechanical or documentation drawings and text, then returning to Layout.

The MAXDXF.EXE file translates a drawing exchange format (.DXF) file and produces an OrCAD binary file with a .MAX extension, or translates a .MAX file to the .DXF format.

Note: Translation between .MAX and .DXF files requires that the file MAXDXF.INI be in the LAYOUT directory.

Note: If you are merging multiple DXF files with data in your .MAX file, do not select the Remove existing obstacles and text option in the DXF Import dialog box. If you want to replace the board outline and its annotations with entities from the .DXF file, select the Remove existing obstacles and text option.

Shortcut

Keyboard: ALT, F, E, X

Layout to IDF command (Export)

Use this command to translate Layout files into Intermediate Drawing Format (IDF).

Layout to IDF dialog box

Layout to GenCAD command (Export)

Use this command to translate Layout files into GenCAD.

Layout to IPC-356 command (Export)

Use this command to create IPC-356 netlists for Layout files.

Layout to Hyperlynx command (Export)

Use this command to translate Layout files into HyperLynx format.

Layout to SPECCTRA command (Export)

Use this command to translate Layout files into SPECCTRA format.

Layout to GenCAM command (Export)

Use this command to translate Layout files into GenCAM format.

Related topics

Layout to GenCAM dialog box

Layout to ODB++ command (Export)

Use this command to translate Layout board files (.MAX) and OrCAD ASCII format files (.MIN) to the Valor ODB++ or XML++ format.

Note: Before running Layout to ODB++ export, you must install the ODB++ Gateway to ODB++ Translator. You can download the ODB++ Gateway to ODB++ Translator from:

http://www.orcad.com/community.layout.dl.aspx

Related topics

ODB++ Export

Layout to ODB++ dialog box

1, 2, 3, 4 file commands (File menu)

Use the numbers listed at the bottom of the File menu to open one of the last four projects or files. Choose the file you want to open.

Shortcut

Keyboard: ALT, F, n (n = 1, 2, 3, or 4)

View Menu Commands (LSession)

Toolbar command (View menu) on page 864

Status Bar command (View menu) on page 864

Toolbar command (View menu)

Use this command to show or hide the toolbar. This setting is stored in your .INI file and thus affects the visibility of the toolbar in subsequent sessions. You can move the toolbar anywhere on the screen by pressing the left mouse button over the toolbar, and then moving the mouse to the toolbar's new location. If the toolbar is moved to any edge of the window, the toolbar snaps into place. Otherwise, it floats on the screen wherever it is released.

If the toolbar is floating, you can hide it by clicking on its left control box.

This command is only available in LSESSION.EXE.

Shortcut

Keyboard: ALT, V, T

Status Bar command (View menu)

Use this command to show or hide the status bar. This setting is stored in your .INI file and thus affects the visibility of the status bar in subsequent sessions.

This command is only available in LSESSION.EXE.

Shortcut

Keyboard: ALT, V, S

Tools Menu Commands (LSession)

Library Manager command (LSession: Tools menu) on page 865

Create (Catalog) command on page 866

Print (Catalog) command on page 866

Define Stackup command on page 866

Configure Design Library command on page 867

OrCAD Capture command (Tools menu) on page 867

IntelliCAD command on page 867

SmartRoute command (Tools menu) on page 868

GerbTool command (Tools menu) on page 868

ECOs command (Tools menu) on page 869

Edit App Settings command (Tools menu) on page 880

Reload App Settings command (Tools menu) on page 881

Library Manager command (LSession: Tools menu)

Note: **Running multiple copies of Layout is not recommended.** When multiple copies of Layout are running, data is shared between the two copies. Performing processes like AutoECO or making edits in the Library Manager can cause changes in other open copies.

Use this command to bring up Layout's library manager. Using Layout’s library manager, you can:

- view any footprint of any library

- edit, create, or delete a footprint
- make libraries available for use with your design
- create custom libraries.

Shortcut

Keyboard: ALT, T, L

Managing libraries and footprints

Create (Catalog) command

The Create command launches the Create Catalog dialog box. This dialog allows you to create Catalog files from Layout library (.LLB) or board (.MAX) files.

Catalog files are .MAX files that have all components displayed in rows and divided into pages. The number of catalog pages contained in each MAX file varies with the size of the final printed page. Typically about six size A (letter) catalog pages will fit in each MAX file. The Catalog tool creates as many .MAX files as are necessary to catalog the library file.

Print (Catalog) command

The Print command launches the Print Catalog dialog box. This dialog box allows you to print Catalog .MAX files that you have already created with the Catalog tool.

The Print command is located in the session frame and can be found on the Tools menu. Click Tools, point to Catalog and choose Print.

Define Stackup command

The Define Stackup command launches the Create Stackup dialog box. You can use this dialog box to specify the path to

a predefined technology file on which you want base the new stackup.

The stackup information from the selected technology file is displayed in the Stackup Editor dialog box. For more information, see Stackup Editor dialog box.

Shortcut

Keyboard: ALT, T, K

Configure Design Library command

The Configure Design Library command launches the Configure Design Library dialog box. You can use this dialog box to create design libraries— libraries that contain all the footprints required for a design. For more information, see Creating design libraries on page 570.

Shortcut

Keyboard: ALT, T, D

OrCAD Capture command (Tools menu)

Use this command to run OrCAD Capture for Windows. This is useful for cross probing with your schematic designs.

Shortcut

Keyboard: ALT, T, C

IntelliCAD command

Use the IntelliCAD command to launch IntelliCAD.

To launch IntelliCAD

1 In the session frame, from the Tools menu, select IntelliCAD.

SmartRoute command (Tools menu)

Use this command to run SmartRoute, a shape-based, gridless autorouter. SmartRoute features easy setup, fast routing speeds, high completion rates, and high quality routing.

Shortcut

Keyboard: ALT, T, S

GerbTool command (Tools menu)

Use this command to read and write all standard Gerber formats and IPC-356. This provides features for automatic tear-dropping, panelization, venting and thieving, and removal of unused pads and silkscreen on pads. These functions are all required for optimal manufacturability

To launch GerbTool

1 In the session frame, from the Tools menu, point to GerbTool and choose:

New command (Tools menu, GerbTool command)

or

Open command (Tools menu, GerbTool command)

Shortcut

Keyboard: ALT, T, G

New command (Tools menu, GerbTool command)

Use this command to create a new GerbTool Layout design file (*.GTD).

Shortcut

Keyboard: ALT, T, G, N

Open command (Tools menu, GerbTool command)

Use this command to open and modify an existing GerbTool Layout design file (*.GTD).

Shortcut

Keyboard: ALT, T, G, O

ECOs command (Tools menu)

Use the ECO commands to forward annotate a printed circuit board from OrCAD Capture for Windows (or another vendor's schematic design application) to OrCAD Layout for Windows.

AutoECO selectively communicates information from Capture to Layout, or from one printed circuit board to another. In other words, when bringing data into Layout from Capture, you can choose the appropriate AutoECO option to annotate only the schematic data that has changed. This way, you can avoid inadvertently overriding board data that you do not want to change. You can also choose from two AutoECO options that communicate information between printed circuit boards.

The first six commands in the menu transfer data from Capture to Layout during forward annotation. The last command, AutoECO/Net Attrs, communicates information between printed circuit boards.

- AutoECO – the basic AutoECO. It adds and deletes parts and nets but it does not change footprints, part locations or net properties.
- AutoECO/Override Attrs – adds and deletes parts and nets, overrides net properties and part properties. It changes footprints but does not change locations of existing parts. Components that are no longer needed are deleted.
- AutoECO/Override Coords – merges new components and connections with an existing board. It overrides all of the placement coordinates in an existing board, and reconciles all components and nets with the netlist.
- AutoECO/Override All – merges new components and connections with an existing board. This ECO reconciles all components, nets and properties with the netlist, and also moves all components to the netlist coordinates. Components and nets that are no longer needed are deleted.
- AutoECO/Add Only – adds nets and components, but does not change anything that exists on the board. This option is useful when the board is near completion and you wish to add some parts (for example, a bypass capacitor). Properties on existing components are not updated.
- AutoECO/Add Override – adds new components and nets, and overrides component and net properties. Unused components and nets are not deleted.
- AutoECO/Net Attrs – transfers net attributes such as width, weight, spacing-per-layer, width-per-layer, and reconnection type from one board to another.

Shortcut

Keyboard: ALT, T, O

The ECO commands are a function of the session window.

To access the ECO commands

1. Open the session window.
2. From the Tools menu, point to ECOs and choose the AutoECO command of your choice. The AutoECO dialog box appears.

Note: The ECO you chose from the menu is preset in the dialog box. ECO settings can be changed in the dialog, and are not constrained by your original menu choice.

Related topics

For more information about	See this topic
Basic ECO command description	ECOs command
The AutoECO command	AutoECO command
The AutoECO / Override Attributes command	AutoECO / Override Attrs command
The AutoECO / Override Coordinates command	AutoECO / Override Coords command
The AutoECO / Override All command	AutoECO / Override All command
The AutoECO / Add Only command	AutoECO / Add Only command
The AutoECO / Add Override command	AutoECO / Add Override command
The AutoECO / Net Attributes command	AutoECO / Net Attrs command

AutoECO command (Tools menu, ECOs command)

AutoECO adds and deletes components and nets, but does not override board attributes. This version is executed when Layout runs AutoECO automatically.

The following table shows the effect on the various elements of the destination board.

Destination board element	AutoECO MNL to MAX
Components	Match schematic

Destination board element	AutoECO MNL to MAX
Nets	Match schematic
Obstacles	No changes
Text	No changes
Placement	No changes except new
Component Properties except Coordinates	No changes except new
Component Coordinates	No changes except new
Net Properties	No changes except new

Match schematic

All existing data on the PCB in this category is made to match the schematic page, including additions and deletions.

No changes except new

All existing data on the PCB in this category remains the same, except additions are made from the schematic page.

No changes

All data in this category in PCB remains the same.

Shortcut

Keyboard: ALT, T, O, E

Related topics

AutoECO/Override Attrs command (Tools menu, ECOs command)

AutoECO/Override Attrs merges new components and connections with an existing board. It adds and deletes parts and nets, overrides net properties and part properties. It changes footprints but does not change locations of existing parts. Components that are no longer needed are deleted.

This command should not be used for creating new boards.

The following table shows the effect on the various elements of the destination board.

Destination board element	AutoECO / Override Attrs MNL to MAX
Components	Match schematic
Nets	Match schematic
Obstacles	No changes
Text	No changes
Placement	No changes
Component Properties except Coordinates	Match schematic
Component Coordinates	No changes except new
Net Properties	Match schematic

Match schematic

All existing data on the PCB in this category is made to match the schematic page, including additions and deletions.

No changes except new

All existing data on the PCB in this category remains the same, except additions are made from the schematic page.

No changes

All data in this category in PCB remains the same.

Shortcut

Keyboard: ALT, T, O, A

AutoECO/Override Coords command (Tools menu, ECOs command)

AutoECO/Override Coords merges new components and connections with an existing board. It overrides all of the placement coordinates in an existing board, and reconciles all components and nets with the netlist.

This command should not be used for creating new boards.

The following table shows the effect on the various elements of the destination board.

Destination board element	AutoECO MNL to MAX
Components	Match schematic
Nets	Match schematic
Obstacles	No changes
Text	No changes
Placement	Match schematic
Component Properties except Coordinates	No changes except new

Destination board element	AutoECO MNL to MAX
Component Coordinates	Match schematic
Net Properties	No changes except new

Match schematic

All existing data on the PCB in this category is made to match the schematic page, including additions and deletions.

No changes except new

All existing data on the PCB in this category remains the same, except additions are made from the schematic page.

No changes

All data in this category in PCB remains the same.

Shortcut

Keyboard: ALT, T, O, C

Related topics

AutoECO/Override All command (Tools menu, ECOs command)

AutoECO/Override All merges new components and connections with an existing board. This ECO reconciles all components, nets and properties with the netlist, and also moves all components to the netlist coordinates. Components and nets that are no longer needed are deleted.

This command should not be used for creating new boards.

The following table shows the effect on the various elements of the destination board.

Destination board element	AutoECO MNL to MAX
Components	Match schematic
Nets	Match schematic
Obstacles	No changes
Text	No changes
Placement	Match schematic
Component Properties except Coordinates	Match schematic
Component Coordinates	Match schematic
Net Properties	Match schematic

Match schematic

All existing data on the PCB in this category is made to match the schematic page, including additions and deletions.

No changes

All data in this category in PCB remains the same.

Shortcut

Keyboard: ALT, T, O, L

AutoECO/Add Only command (Tools menu, ECOs command)

AutoECO/Add Only adds nets and components, but does not change anything that exists on the board. This option is useful when the board is near completion and you wish to add some parts (for example, a bypass capacitor). Properties on existing components are not updated.

The following table shows the effect on the various elements of the destination board.

Destination board element	AutoECO MNL to MAX
Components	Match schematic except deletes
Nets	Match schematic except deletes
Obstacles	No changes
Text	No changes
Placement	No changes except new
Component Properties except Coordinates	No changes except new
Component Coordinates	No changes except new
Net Properties	No changes except new

Match schematic except deletes

All existing data on the PCB in this category is made to match the schematic page, including additions, but not deletions.

No changes except new

All existing data on the PCB in this category remains the same, except additions are made from the schematic page.

No changes

All data in this category in PCB remains the same.

Shortcut

Keyboard: ALT, T, O, O

AutoECO/Add Override command (Tools menu, ECOs command)

AutoECO/Add Override adds components and nets and updates component and net properties. It also won't delete components or nets.

The following table shows the effect on the various elements of the destination board.

Destination board element	AutoECO MNL to MAX
Components	Match schematic except deletes
Nets	Match schematic except deletes
Obstacles	No changes
Text	No changes
Placement	Match schematic except deletes
Component Properties except Coordinates	Match schematic except deletes

Destination board element	AutoECO MNL to MAX
Component Coordinates	Match schematic except deletes
Net Properties	Match schematic except deletes

Match schematic except deletes

All existing data on the PCB in this category is made to match the schematic page, including additions, but not deletions.

No changes

All data in this category in PCB remains the same.

Shortcut

Keyboard: ALT, T, O, V

AutoECO/Net Attrs command (Tools menu, ECOs command)

AutoECO/Net Attrs transfers net attributes such as width, weight, spacing-per-layer, width-per-layer, and reconnection type from one board to another.

The following table shows the effect on the various elements of the destination board.

Destination board element	AutoECO MAX to MAX
Components	No changes
Nets	No changes
Obstacles	No changes

Destination board element	AutoECO MAX to MAX
Text	No changes
Placement	No changes
Component Properties except Coordinates	No changes
Component Coordinates	No changes
Net Properties	Match original board where possible

No changes

All data in this category in PCB remains the same.

Match original board where possible

Any matches to existing data on the old PCB are replaced from new PCB design, but no additions or deletions are made.

Note: You may substitute another MAX file where the AutoECO program calls for an .MNL file. The components, netlist, and properties will pass forward from the MAX file exactly as if you had started with an .MNL file.

Shortcut

Keyboard: ALT, T, O, N

Edit App Settings command (Tools menu)

Use this command to edit your Layout application settings. After you edit these settings, reload them using the Reload App Settings command (Tools menu).

Shortcut

Keyboard: ALT, T, E

Reload App Settings command (Tools menu)

Use this command to reload your Layout application settings after you have edited them with the Edit App Settings command (Tools menu).

Shortcut

Keyboard: ALT, T, R

Layout Commands

20

File Menu Commands on page 883

Edit Menu Commands on page 895

View Menu Commands on page 901

Tool Menu Commands on page 909

Options Menu Commands on page 973

Auto Menu Commands on page 984

Window Menu Commands on page 1003

Help Menu Commands on page 1005

File Menu Commands

New command (File menu) on page 884

Open command (File menu) on page 885

Load command (File menu) on page 885

Save command (File menu) on page 889

Save As command (File menu) on page 889

Backup command (File menu) on page 892

Close command (File menu) on page 893

Print/Plot command (File menu) on page 893

Library Manager command (File menu) on page 894

Text Editor command (File menu) on page 895

Exit command (File menu) on page 895

New command (File menu)

By choosing the New command on the File menu, you can easily create a new board from an .MNL file (a Layout binary netlist), which is, in turn, generated from a schematic capture design. You can do the same thing by choosing the New Design command on the Tools menu of the session frame. When you want to translate a board from another CAD system, you use one of the Import commands rather than this command.

If you create a netlist from a schematic capture application, you will have created an MNL file (a Layout binary netlist). If you create a new PCB offline, run the AutoECO utility first, then load the new PCB into Layout.

When you choose the New command, Layout first prompts you for a technology template, which contains either a blank board, or might contain board outlines and fixed components, if you have created your own custom file.

Layout then prompts you for an MNL file. You can create an .MNL file in OrCAD's Capture for Windows by choosing Create Netlist from the Tools menu, then choosing the Layout tab.

Finally, Layout executes AutoECO, which combines the electrical information from the MNL file with the physical information of the technology template.

Shortcuts

Keyboard: ALT, F, N

CTRL+N

Open command (File menu)

The Open command presents a list of all .MAX files present on the current directory. You can choose a filename by either selecting a name on the list or entering the name (and directory path, if different from your current directory).

To accept a filename from the File List, select the name and then click OK or press ENTER.

If you want to load a new file over an existing design, Layout asks you whether you want to save any current changes. After you select either YES to save or NO to discard, the Open dialog box appears.

Use the scroll bar in the File Name box to view any additional ".MAX" files in the current directory.

Shortcuts

Keyboard: ALT, F, O

CTRL+O

Load command (File menu)

Use the Load command to load the following:

- Load Strategy command (File menu)
- Load Template File command
- Load Merge board command
- Load Color Setup command
- Load Post Processing Setup command
- Aperture list

Shortcut

Keyboard: ALT, F, l

Load Strategy command (File menu)

Use this command to select one of the OrCAD-provided strategy files or a file that you have created by saving a board with a .SF extension using the Save As Strategy command.

A routing strategy file determines which default routing layers to use, when to use vias, which direction the track should travel, which colors to use, and the size of the active routing window. A placement strategy file determines the placement of components based on different priorities, such as whether clusters are used, whether gates and pins are to be swapped, or whether you want the fastest placement.

Load Template File command

Use this file to select one of the OrCAD-provided technology templates or a file that you have created by saving a board with a .TCH or .TPL extension.

When you save a board with a .TCH or .TPL extension, then load it in as a technology template, Layout replaces certain settings and ignores others. To summarize: the placement and routing strategies, number of defined layers, layer names, and layer properties (such as spacing) are replaced, as well as the grids, the number of defined vias, and the padstacks. Color, package, footprint, and component information is ignored; and nets, connections, obstacles, text, and all other information not specifically mentioned here is ignored.

With a technology template, you can establish the padstacks, route spacing rules, and grid that suit the level of manufacturing technology you will be using.

For more information and descriptions of the technology templates provided by OrCAD, see Technology Templates.

Load Merge board command

The Merge board command is most useful for placing components and merging the routing of a master board section into a design.

All of the information necessary to duplicate a section or template from one design to another is merged: grids, spacing, padstacks, package, footprints, components, nets, tracks, copper, free track, detail (such as obstacles), and text.

When you perform the Merge command, you are prompted: "If any components on the board you're loading have the same reference designator as an existing component, do you want to make a copy or update the existing comp? Answer YES for copy. Answer NO for update."

If you answer Yes, components in the merging file with the same reference designators as components in the open design are added to the open design, with three zeros appended to their names. For instance: U1 becomes U1000, R34 becomes R34000. The original components in the open design retain their reference designators.

If you answer No, components in the open design are replaced by components with the same reference designator from the merging file.

Conventions

In the following descriptions, the term *open design* refers to the currently opened design into which a .MAX file is being merged. The term *merging file* refers to the .MAX file being merged into the open design.

The following conventions apply when merging padstack, footprint, package, component, and net information:

- If an object in the merging file has the same name as an object in the open design, the merging file's version is assumed to be the correct version, and replaces the open design object. If the object has a different name, it is assumed to be an additional object and is added to the open design.

- When objects from the merging file replace objects in the open design, all pertinent information comes from the merging file (placement, rotation, board side, and so on).
- Master footprints in the merging file replace copies of master footprints in the open design.

Note: It is not recommended that the merge board feature be used to change a footprint in the open design unless the pin-out is identical If the pin-out is not identical, nets will be tied together that should not be.

- Nets of the same name are assumed to be the same net. Net properties from the merging file replace the corresponding properties of the open design net. For instance, if the routed version of a net in the merging file has mixed widths, these are passed to the open design.
- Tracks belonging to a net in the merging file are automatically connected by a rat to the tracks of the same net in the open design. Nets that do not exist in the open design are added.
- Text and obstacles in the merging file are compared to the text and obstacles in the open design. If there are duplicates, one is discarded so duplicate objects are not created.

Load Color Setup command

Use this command to load a new color setup and display it. This command is available when the Color spreadsheet is active.

Load Post Processing Setup command

Use this command to load a binary file that adds to the current post processing setup. This command is available when the Post Process spreadsheet is active

Save command (File menu)

Use this command to save the current design file in the design directory.

Shortcuts

Keyboard: ALT, F, S

CTRL+S

Save As command (File menu)

Use this command to save your design under a new file name and directory.

You can save the file under the suggested name by choosing the OK button. You can change the name by typing in a new name, and you can change the file type to one of the following:

- Save As Layout v7.0 Board
- Save As Library
- Save As Template
- Save As Strategy command
- Save As Post Processing Setup command
- Save As Color Setup command
- Save As Aperture List
- Save As No Extension Records

If you assign an extension other than the default extension for the file type you are saving as, the extension you assign and the default extension will be in the filename. For instance, if you were to save a Layout design named Design1 with a .STD extension, the filename would be listed as Design1.STD.MAX.

If you do not specify an extension, Layout defaults to .MAX.

When the name is correct, choose the OK button. You may cancel the command by choosing the Cancel button.

Shortcut

Keyboard: ALT, F, A

Save As Layout v7.0 Board

Use this command to save the current design file in the v7.00 file format.

If you assign an extension other than the default extension for the file type you are saving as, the extension you assign and the default extension will be in the filename. For instance, if you were to save a Layout design named Design1 with a .STD extension, the filename would be listed as Design1.STD.MAX.

If you do not specify an extension, Layout defaults to .MAX.

Save As No Extension Records

With the Save As command, you can save your .MAX file without any extension records. This allows the file to be read by older versions of Layout.

To save a board file without extension records

1. From the file menu choose Save As. The Save As dialog appears.
2. Name your file and choose No Extension Records from the Save as type drop down box.
3. Click Save.

Save As Aperture List

If you need a specific aperture list from a board, you can easily create one with the Save As command. This creates only an aperture list, and excludes all other data in the .MAX file.

To create an aperture list from the current board file

1. From the File menu choose Save As. The Save As dialog appears.
2. Name your file and choose the Aperture List from the Save as type drop down box.
3. Click Save

Save As Strategy command

Use this command to save your own strategy file (extension .SF) if you find that some variation of a standard OrCAD-supplied strategy file results in more success on your typical designs. If you save the strategy file as STD.SF, it will be automatically loaded into a board design as the board is translated into Layout's binary format. You can also load STD.SF and use it with boards that are not translated.

A routing strategy file determines which default routing layers to use, when to use vias, which direction the track should travel, which colors to use, and the size of the active routing window. A placement strategy file determines the placement of components based on different priorities, such as whether clusters are used, whether gates and pins are to be swapped, or whether you want the fastest placement.

If you assign an extension other than the default extension for the file type you are saving as, the extension you assign and the default extension will be in the filename. For instance, if you were to save a Layout design named Design1 with a .STD extension, the filename would be listed as Design1.STD.MAX.

If you do not specify an extension, Layout defaults to .MAX.

Save As Color Setup command

Use this command to save the current screen colors to a file.

If you assign an extension other than the default extension for the file type you are saving as, the extension you assign and the default extension will be in the filename. For instance, if

you were to save a Layout design named Design1 with a .STD extension, the filename would be listed as Design1.STD.MAX.

If you do not specify an extension, Layout defaults to .MAX.

Save As Post Processing Setup command

Use this command to save a binary file that can be used to transfer a post processing setup from one board to another.

If you assign an extension other than the default extension for the file type you are saving as, the extension you assign and the default extension will be in the filename. For instance, if you were to save a Layout design named Design1 with a .STD extension, the filename would be listed as Design1.STD.MAX.

If you do not specify an extension, Layout defaults to .MAX.

Backup command (File menu)

The Backup command saves your file to BACKUP1.MAX.

In order to avoid the possibility of losing a valuable file, there are always five levels of backup files present in your working directory.

Whenever you choose Backup from the File menu, the old BACKUP1.MAX is copied to BACKUP2.MAX; BACKUP2.MAX is copied to BACKUP3.MAX; BACKUP3.MAX is copied to BACKUP4.MAX; BACKUP4.MAX is copied to BACKUP5.MAX. The old BACKUP5.MAX is discarded.

Whether in autorouting or manual routing, an automatic backup is made by Layout whenever a specified amount of time passes. You specify the amount of time between automatic backups by choosing Backup Interval from the Options menu.

When autorouting, Layout also writes to BACKUP1.MAX each time it switches to a new routing window, provided that some change was made to the previous window.

Shortcut

Keyboard: ALT, F, B

Close command (File menu)

Use the Close command to close a design, but leave Layout open for starting a new design, or opening an existing one.

Shortcut

Keyboard: ALT, F, C

Print/Plot command (File menu)

The Print command displays the Print/Plot dialog box if the graphic editor window is active, or prints the selected portion of the spreadsheet if a spreadsheet window is active.

In order to print in Layout, you must install a printer using the Windows setup program (refer to your Windows documentation for further information on installing and adding printers). In Layout, both printing and plotting are governed by the Windows Print Manager.

Use the Print/Plot command to:

- print to the print manager
- create a DXF file
- create a print (.PRN) file

Note: Customers have reported that according to the Product Comparison Guide for HP-GL/2 and HP RTL Peripherals published by Hewlett-Packard, the following devices are obsolete and no longer supported. Therefore, you may be unable to acquire a current, working driver for the following output devices.

- HP DesignJet plotters (except 200, 600, and 650C)
- HP PaintWriter XL printer

- HP DraftMaster Series plotters - SX, RX, and MX (except the SXplus, RXplus, and Mxplus)
- HP 7600 Series plotters - Models 240D/E, 250, 255, and 355

Certain companies (for example, WinLINE) produce drivers. You can contact WinLINE on the internet at http://www.winline.com.

Note: Windows users need to be sure that their printer is listed as active, and is the default, or Layout will not be able to access it. If these fields are not set properly, you will receive the message "Failed to Init Printer."

Note: To print a Query window, first save the information to a file using Output from the Edit menu. You can then print the file using the standard Print option from the File menu in the Windows File Manager.

Shortcuts

Keyboard: ALT, F, T

CTRL+P

Library Manager command (File menu)

Using the library manager, you can:

- view any footprint of any library
- edit, create, or delete a footprint
- make libraries available for use with your design
- create custom libraries.

Shortcut

Keyboard: ALT, F, I

CTRL+I

Note: **Running multiple copies of Layout is not recommended.** When multiple copies of Layout are running, data is shared between the two copies. Performing processes like AutoECO or making edits in the Library Manager can cause changes in other open copies.

Text Editor command (File menu)

Use this command to open a report file, INI file, or LOG file in a text editor. The text editor is specified in the REPORTEDITOR section of the LAYOUT.INI file.

Shortcut

Keyboard: ALT, F, E

Exit command (File menu)

Use the Exit command to exit Layout. If you have changed the file since the last time you chose Backup or Save As, the program asks you to save your file.

You can also exit this or any other Windows program by double clicking the minimize button in the upper right corner of the window, or by pressing ALT+F4.

Shortcut

Keyboard: ALT+F+X

Edit Menu Commands

Undo command (Edit menu) on page 896

Copy command on page 896

Paste command on page 897

Delete command on page 897

Find/Goto command (Edit menu) on page 897

Select Any command (Edit menu) on page 898

Select Next command (Edit menu) on page 899

Clear Selections command (Edit menu) on page 900

End command (Edit menu) on page 900

Properties command (Edit menu) on page 901

Undo command (Edit menu)

Use this command to reverse the effect of the last operation. The name of this command changes, depending on the last reversible operation.

If the last operation is not reversible, the command is not available.

Shortcuts

Keyboard: ALT, E, U

CTRL+Z

CTRL+U

Copy command

Use this command to copy an object and have Layout place it on the pointer for placement.

Shortcuts

Keyboard: ALT, E, C

CTRL+C

Paste command

Use this command to paste copied objects into the design.

Shortcuts

Keyboard: ALT, E, P

CTRL+V

Delete command

The edit Delete command deletes a single object or multiple objects.

To delete components

1. Single or multiple select the object(s) you want to delete.
2. After the selection, do one of the following:

- Click the Delete button on the toolbar.
- Press the DELETE key on the keyboard.
- From the Edit menu, choose Delete.

Shortcuts

Keyboard: ALT, E, D

CTRL+X

Find/Goto command (Edit menu)

The Find/Goto command lets you find objects by name or location. You can also place objects with this command.

To find a component

- Use the form U1

To find a component and pin number

- Use the form U1.1

To find a coordinate

- Use one of these forms:
  1000,1000
  or
  1.000,1.000

To place an object

- While you are dragging a component, obstacle, or text item, you can enter a set of coordinates into the Find dialog box, and Layout will automatically place your object, or obstacle vertex, or text object in the location you have specified. This feature is available even if the location is off-grid.

Shortcuts

Keyboard: ALT, E, F

CTRL+F

Select Any command (Edit menu)

Select Any is available when you have the Component tool selected. Select automatically displays the Component Selection Criteria dialog box and lets you select a group of components fitting certain criteria. The default criteria is all components except those that are fixed.

For example, if you select all components beginning with C, components fitting this criteria appear on your pointer and you can build a cluster or Quick Place the components.

If you want to select a group of components one at a time, use the Queue for Placement command (Component)

Criteria

- Reference Designator
- Footprint Name
- Number of pins
- Comp Group
- Placed
- Locked

Shortcuts

Keyboard: ALT, E, S

ALT+S

Select Next command (Edit menu)

Select Next is available when you have the component tool, the error tool, or one of the routing tools selected. It automatically finds and selects the next unplaced component for modification or placement, finds and selects the next error for modification or completion, or finds and selects the next unrouted connection that is DRC activated for modification or completion.

Components

The selection order for Select Next is as follows: If you have selected a cluster, the key component (see the section on Key Components) from that cluster is picked first, then the rest of the components in that cluster.

Components within non-selected clusters are picked next, starting again with any key components; any other unplaced components are selected after all existing clusters have been emptied.

To override, choose Select Next and select the component you wish to place from the list provided. If there are too many

components to list, you get a dialog box where you can enter the name of the component you wish to place.

Errors

With the error tool selected, Select Next moves to the next sequential error and highlights it.

Routing tools

You normally use this command at the end of a design to find the last few connections without having to visually scan the board.

Shortcuts

Keyboard: ALT, E, N

N

Clear Selections command (Edit menu)

Use this command to clear all selected items in a spreadsheet.

Shortcut

Keyboard: ALT, E, C

End command (Edit menu)

End Command deselects any selected objects, and exits the active command mode.

Shortcuts

Keyboard: ALT, E, E

Properties command (Edit menu)

When you choose the Properties command, after selecting an object, Layout displays a dialog box, in which you can edit the object's properties.

Object selected	Dialog box
■ none	Edit Layer
■ component	Edit Component
■ group	Edit Component
■ obstacle	Edit Obstacle
■ pin	Modify Connections
■ text	Edit text

Shortcuts

Keyboard: ALT, E, P

CTRL+E

View Menu Commands

Design command (View menu) on page 902

Density Graph command on page 902

Preview command (View menu) on page 904

High Contrast command (View menu) on page 904

Clear Screen command (View menu) on page 904

Redraw command (View menu) on page 905

Query Window command (View menu) on page 905

Database Spreadsheets command (View menu) on page 906

Zoom All (Fit) command (View menu) on page 906

Zoom Center command (View menu) on page 907

Zoom In command (View menu) on page 907

Zoom Out command (View menu) on page 908

Zoom Previous command (View menu) on page 908

Zoom DRC/Route Box command (View menu) on page 909

Visible <> Invisible command (View menu) on page 909

Design command (View menu)

Use this command to return to the design window from the density graph and preview windows.

When returning to the design window from the preview window, this command reverses the two-color plot image of a layer, with the color image of the layer as it appeared before the preview.

Shortcut

Keyboard: ALT, V, D

Density Graph command

Point to this View menu item to display the density graph options. Choose from the three options (coarse, medium or fine) to display the density graph.

Fine command on page 902

Medium command on page 903

Coarse command on page 903

Fine command

This command causes Layout to display the density graph at the highest level of detail. Screen drawing is slower at this level than at the other two levels.

You might want to use the Fine density graph representation after you have optimized your layout with the Coarse command or Medium command. Optimization at this level usually targets particularly troublesome areas of the layout.

Shortcuts

Keyboard: ALT, V, H, F

Medium command

This command causes Layout to display the density graph at an intermediate level of detail. If you find that redrawing at the Fine level is too slow, try working at the Medium level.

You may want to use the Medium density graph representation after you have optimized your layout with the Coarse command, but before you have optimized with the Fine command.

Shortcuts

Keyboard: ALT, V, H, M

Coarse command

This command causes Layout to display the density graph at a low level of detail. Layout redraws more quickly at this level than at the other two.

You may want to use the Coarse density graph representation for your first optimization pass. This allows you to address layout in areas that are likely to prove troublesome early. In subsequent optimization efforts, use the Medium command and Fine command.

Shortcuts

Keyboard: ALT, V, H, R

Preview command (View menu)

This command displays a "picture" of the plot file that will be produced with the current settings. The preview allows you to visually inspect the contents to ensure that all the information you need is included in the plot file.

To return to the design window after previewing the plot file, choose Design from the View menu.

Note: If the background remains black, instead of changing to white, when you choose Preview from the Post Process spreadsheet's pop-up menu, choose Redraw from the View menu.

Shortcuts

Keyboard: ALT, V, W

High Contrast command (View menu)

This command is a toggle that isolates the currently selected object. When high contrast is turned on, all objects, except for the selected object, change to shades of gray. The selected object retains its original color so that it is highly visible.

Choose the High Contrast command again to turn off the high contrast display.

Shortcuts

Keyboard: ALT, V, T

(period)

Clear Screen command (View menu)

Use this command to erase all data from the screen without affecting the routing of the board.

If you want to view only a selected layer, you can erase the screen then simply press the number associated with that layer in the layers drop-down list of the design window. For instance, if you want to view the top layer alone, you would choose Erase Screen from the View menu, the press the key 1.

Shortcuts

Keyboard: ALT, V, S

BACKSPACE

Redraw command (View menu)

Redraw lets you repaint the screen.

Shortcuts

Keyboard: ALT, V, R

HOME

Query Window command (View menu)

The Query Window command lets you query Layout for more information on the item currently selected in the design or spreadsheet window.

There is a default Query window present when you start Layout that you can use by clicking the Query button on the toolbar.

By selecting any object, such as a pad, track, obstacle or connection, you can find out various kinds of information, such as the name of an object (for example, U1.1), its location, its attached signal net name (such as GND), and any routing attributes attached to the signal.

More importantly, you can use Find, Edit, or Delete with any object in the design without having to active a data or graphics window. In addition, by clicking the left mouse button on any

quoted string, you can initiate a Query for the object, and modify it or inspect the data as you wish.

If you click on a location (for example, [1000,1000]) the graphics window zooms to the new location.

Shortcuts

Keyboard: ALT, V, Q

Q

Database Spreadsheets command (View menu)

Use this command to display the database spreadsheet windows. Database spreadsheet windows contain textual data, similar to that found in a Query window. The difference is that the spreadsheets are laid out so that you can easily edit many items at a time, rather than just one at a time as in a Query window.

You can display a dialog box for editing spreadsheet information by double-clicking on a cell. To select multiple cells, press and hold SHIFT while you select the cells.

Shortcuts

Keyboard: ALT, V, D

Zoom All (Fit) command (View menu)

Zoom All (Fit) lets you quickly display the entire board at once, with as much detail as the screen can display.

Note: For the Zoom commands to work properly in half screen mode, you must reset the design window after choosing half screen. To do so, choose the Reset All command from the Window menu.

Shortcuts

Keyboard: ALT, V, A

SHIFT+HOME

Zoom Center command (View menu)

Zoom Center lets you center the current position of the pointer on the screen, without changing the aspect ratio of the screen.

Note: For the Zoom commands to work properly in half screen mode, you must reset the design window after choosing half screen. To do so, choose the Reset All command from the Window menu.

Shortcuts

Keyboard: ALT, V, C

C

Note: You can pan from one area of the design to another by holding down the C key and moving the mouse.

Zoom In command (View menu)

Zoom In lets you see a smaller area of the board in greater detail.

Zoom In increases the scale of the board by 2:1 each time you click the left mouse button.

You can also use Zoom In to zoom to an area, to isolate an area of the board for routing or inspection, without changing the size of the DRC/Route box.

When you select Zoom In, the pointer changes to a Z.

Note: For the Zoom commands to work properly in half screen mode, you must reset the design window after choosing half screen. To do so, choose the Reset All command from the Window menu.

Shortcuts

Keyboard: ALT, V, I

I

Zoom Out command (View menu)

Zoom Out lets you view a larger area of the board in less detail.

Zoom Out decreases the scale of the board by 2:1 each time you click the left mouse button.

Press ESC to get out of Zoom mode.

Note: For the Zoom commands to work properly in half screen mode, you must reset the design window after choosing half screen. To do so, choose the Reset All command from the Window menu.

Shortcuts

Keyboard: ALT, V, O

O

Zoom Previous command (View menu)

Zoom Previous lets you display the board as it was displayed previously to the last use of Zoom.

Note: If you use Zoom/DRC/Route box, you will not be able to use Zoom/Previous to go back to your last location or graphics aspect ratio. If you wish to inspect a certain area of the design and then go back to where you were, you should use Zoom/Area instead.

Note: For the Zoom commands to work properly in half screen mode, you must reset the design window after choosing half screen. To do so, choose the Reset All command from the Window menu.

Shortcut

Keyboard: ALT, V, P

Zoom DRC/Route Box command (View menu)

The DRC/Route box command has two functions:

- Use the pointer to move the current sized DRC/Route box and zoom in.
- Resize the DRC/Route box with the mouse and zoom in to it.

Note: For the Zoom commands to work properly in half screen mode, you must reset the design window after choosing half screen. To do so, choose the Reset All command from the Window menu.

Shortcuts

Keyboard: ALT, V, B

B

Visible <> Invisible command (View menu)

Use this command to set the selected item's color to invisible. An invisible item is not displayed.

Shortcut

Keyboard: ALT, E, V

Tool Menu Commands

Layer Flyout Commands on page 910

Block command on page 911

Cluster Flyout Commands on page 916

Group Flyout commands on page 917

Matrix Flyout Commands on page 921

Component Flyout Commands on page 923

Gate Flyout Commands on page 930

Footprint Flyout Commands on page 930

Padstack Flyout Commands on page 932

Pin Flyout Commands on page 935

Aperture Flyout Commands on page 938

Net Flyout Commands on page 939

Connection Flyout Commands on page 944

Track Flyout Commands on page 947

Track Segment Flyout Commands on page 950

Jumper Flyout Commands on page 952

Via Flyout Commands on page 954

Test Point Flyout Commands on page 958

Drill Chart Flyout Commands on page 960

Text Flyout Commands on page 960

Dimension Flyout Commands on page 964

Measurement Flyout Commands on page 966

Obstacle Flyout Commands on page 967

Error Flyout Commands on page 972

Layer Flyout Commands

Select Layer command on page 911

Unroute command (Layer) on page 911

Properties command (Layer) on page 911

Select Layer command

The Select Layer command lets you paint a particular layer on top of the existing graphics.

When the Select Layer dialog box appears, choose the layer you want to display on top, from the Layer drop-down list.

You can also use the Layer drop-down list on the toolbar.

If you want to see one layer at a time, press BACKSPACE to erase the screen and then select the layer you wish to view.

Shortcut

Keyboard: ALT, V, L

Unroute command (Layer)

Use this command to unroute an entire, previously routed layer.

Properties command (Layer)

After selecting a layer, choosing the Properties command displays the Edit Layer dialog box.

Shortcuts

Keyboard: CTRL, E

Block command

Using the Block command, you can select and manipulate multiple elements in a board file. With the Block command you can select components with any attached obstacles and text,

unattached obstacles, unattached text, and tracks and/or track segments. Text or obstacles attached to components cannot be individually selected using the block move tool. After a selection is created, you can move, rotate, lock, or delete the selected block.

Note: The Online Design Rules Checker does not check design rules when you place a selected block.

Block Command menu items

Select Tool command (Block)	Enables the Block selection tool.
Select All command (Block)	Selects all items on the board except datum, error markers and the drill chart. **Note:** This command is only available when no other item is selected.
Rotate command (Block)	Rotates the currently selected block.
Lock command (Block)	Locks all selected items at the current location
Delete command (Block)	Deletes all selected items.

Additional pop-up menu items

End Command (pop-ups)	Places the current selection and ends use of the Block tool.
Move On/Off command (Block)	When enabled, this command attaches the current selection to the cursor. When disabled, the selection is dropped but not placed in the current location.

Undo command (Edit menu)	Undo the most recent action.

Block selection options

The Block command has three selection options that help you refine the scope of what the Block tool selects. These options are available from the pop-up menu.

Select Entire Track

When checked, a selection picks up the entire track for any net inside the selection bounds. When the selected block is moved, routed tracks maintain the original route structure, and when placed, a ratsnest reconnects broken nets.

When unchecked, a selection cuts any already routed tracks at the border of the selection. Broken nets are reconnected by a ratsnest.

Note: When disabled, you can CTRL-click to add or remove individual track segments to a selection.

Select Locked and Fixed

When checked, all locked and fixed items within the selection bounds are selected and can be manipulated.

Note: This setting does not override the locked status of the objects when manipulating them with other tools.

When unchecked, locked and fixed items are not selected, and cannot be manipulated.

Select from Invisible Layers

When checked, all items on invisible layers within the selection bounds are included in the selection.

Note: Invisible items can be moved and rotated even though the items may remain invisible.

When unchecked, invisible items are not selected, and cannot be manipulated.

Select Tool command (Block)

With the Block tool, you can select and manipulate multiple elements on a board. To enable the Block tool, from the Tool menu, point to Block and choose Select Tool.

Shortcuts

Keyboard: ALT, T, L, ENTER

Select All command (Block)

With the Bock tool enabled, you can select all items on the board except datum, error markers and the drill chart.

To select all items on the board

1. From the Tool menu, point to Block and choose Select Tool. This enables the Block tool.
2. From the Tool menu point to Block and choose Select All. All items on the board highlight, except datum, error markers and the drill chart.

Note: You can only use the Select All command when you have not made any other selections.

Shortcuts

Keyboard: ALT, T, L, A

Pop-up: Select All

Rotate command (Block)

The Rotate command rotates the currently selected block.

To rotate the entire board

1 From the Tool menu point to Block and choose Select All. All items on the board are selected except for datum, error markers and the drill chart.

Note: You can only use the Select All command when you have not made any other selections.

2 From the Tool menu point to Block and choose Rotate.

3 Right-click in the design window and choose End Command from the pop-up menu to place the board.

Note: Placing a rotated board may take several minutes.

Shortcuts

Keyboard: R

Lock command (Block)

The Lock command locks the currently selected block.

Note: With the Select Locked and Fixed option enabled, locked elements can be selected and manipulated with the Block tool.

Shortcuts

Pop-up: Lock

Delete command (Block)

Use the Delete command to delete the currently selected block.

Shortcuts

Keyboard: CTRL X

Move On/Off command (Block)

Use the Move On/Off command to move a selected block.

To move a selection with the Move On/Off command

1. From the Tool menu point to Block and choose Select Tool. This enables the Block tool.
2. Create a rectangular or polygonal selection.
3. Right click on the selection and choose Move On/Off from the pop-up menu. The selection is attached to the pointer.
4. Move the selection to the desired location.

Note: To temporarily drop the selection, right click and choose Move On/Off.

5. Click to place the selection.

Any Angle Corners / 135's and 90's only command (Block tool)

Use this command to choose between creating polygonal block selections with 135 and 90 degree vertices or selections with corners of any angle.

Cluster Flyout Commands

Make command (Cluster) on page 917

Break command (Cluster) on page 917

Make command (Cluster)

Make is available when the Component tool is selected. Use this command to define a component cluster, which is an intuitive way of organizing components into groups in the preliminary stages of a design.

Shortcuts

Keyboard: K

Break command (Cluster)

Break Cluster is available when you have the Component tool selected. Break Cluster restores all of the components in a cluster to their actual shapes. All of the component origins will be located at the center point of the cluster from which they were derived. This is useful if you need to take a quick look inside a cluster, to see what shapes are present.

If you select a cluster and press S, you see a listing that gives you the number of pins in each component. This is often sufficient to identify the types of components you may wish to place first.

Shortcuts

Keyboard: CTRL+K

Group Flyout commands

New command (Group) on page 918

Opposite command on page 918

Rotate command on page 919

Lock command on page 920

Properties command (Group) on page 920

Delete command (Groups) on page 921

New command (Group)

You can assign components as a group while you are working in Capture before you create a netlist, or while you are working in Layout by using the Components spreadsheet. The preferable approach is to assign groups while you are working in Capture.

In Capture, for each component in the group, you need to generate a user-defined part property with property name COMPGROUP and a common integer as the property value. See the Capture documentation for detailed instructions.

If you are working with Layout Plus, consider that clustering components yields quicker placement than grouping does.

To group components in Layout

1. In the Components spreadsheet, select the components that you want to group.
2. From the pop-up menu, choose the Properties command to display the Edit Component dialog box.
3. Assign an integer as a group number and choose the OK button.

Opposite command

Use this command to mirror text, components, groups, and obstacles to the opposite layer. See Layers spreadsheet for a description of the opposite layer.

Single selection of text, components, groups, or obstacles

When you choose the Opposite command, a dotted version of the selected text, component, or obstacle appears on your pointer, mirrored on the X axis, and automatically moved to the opposite layer (according to the mirror layer listed on the Layers spreadsheet).

You can use this command on components whether the Reconn Enabled button of the toolbar is selected or not selected.

Multiple selection of text, components, groups, or obstacles

If you select multiple instances of text, components, groups, or obstacles, and press T for opposite, each instance is mirrored on its own X axis and automatically moved to the opposite layer (according to the mirror layer listed on the Layers spreadsheet).

You can use this command when Reconn Enabled is selected.

Shortcuts

Keyboard: ALT, T, G, T

T

Rotate command

Use this command to rotate components, groups, text, and obstacles. Selected objects rotate according to the increment values specified in the System Settings dialog box. You can rotate objects counterclockwise or clockwise. A positive value creates a counterclockwise rotation, and is the default direction.

Components, groups, and obstacles can be rotated, even when Reconnect mode is on.

Rotating a component, group, obstacle, or text

When you choose the Rotate command, the selected object rotates about the centroid. To continue rotating the object, press R repeatedly.

Rotating multiple components, groups, obstacles, or text

If you select multiple obstacles and rotate them, the relationship between them remains the same. The entire group rotates around the centroid, rather than each object rotating in about its own centroid.

Shortcuts

Keyboard: ALT, T, G, R

R

Lock command

You can temporarily lock a component or group in a specific location.

You can unlock a locked object by selecting it and answering Yes when Layout asks if you want to override the lock.

You can also unlock an object by opening the Components data spreadsheet, selecting the component, opening the Edit Component dialog box, and clearing the Fixed check box.

Shortcuts

Keyboard: ALT, T, G, L

L

Properties command (Group)

After selecting a group, choosing the Properties command displays the Edit Component dialog box.

Shortcuts

Keyboard: CTRL+E

Delete command (Groups)

The group Delete command deletes all components within a selected group.

After selecting a group, there are three ways to activate the Delete command:

- Click the Delete button on the toolbar.
- Press the DELETE key on the keyboard.
- From the Edit menu, choose Delete.
- From the Tool menu, point to Group and choose Delete.

Layout asks for verification that you want to delete all the components within that group. If you continue, Layout notifies you that the netlist will be affected, and verifies again that you want to delete the components.

Shortcuts

Keyboard: CTRL+X

Matrix Flyout Commands

Select Tool command (Matrix) on page 921

Delete command (Matrix) on page 922

Select Tool command (Matrix)

Matrix lets you graphically design a matrix for component placement.

To design a matrix

1. From the Tool menu, point to Matrix and choose Select Tool.
2. Click at a place on your design where you want to start the matrix definition.

3 Drag a rectangle to the size you want your matrix.

4 Click to place the matrix border.

5 Drag inside the matrix border to establish the number of horizontal and vertical cells you want within the matrix.

6 Click again to place the cells.

At this point you can place selected components using the Place Matrix command in the Auto menu. If Layout is unable to fit all of the selected components within the given matrix, the remaining components will be placed in a cluster.

You can also make the following changes to the matrix structure:

- **Move single matrix line.** Click the left mouse button on any matrix to move it; horizontal lines move up or down, vertical lines move left or right.
- **Move entire matrix.** If you use area select to select the matrix, you can move the entire structure in any direction.

Delete command (Matrix)

The matrix Delete command deletes an existing matrix.

To delete a matrix

1 Drag a selection rectangle around the entire matrix.

2 After selecting the matrix, do one of the following:

- Click the Delete button on the toolbar.
- Press the DELETE key on the keyboard.
- From the Edit menu, choose Delete.
- From the Tool menu, point to Matrix and choose Delete.

Shortcuts

Keyboard: CTRL+X

Component Flyout Commands

Select Tool command (Component) on page 923

Select Filtered command (Component) on page 924

Queue for Placement command (Component) on page 924

Place command (Component) on page 924

New command (Component) on page 924

Copy command (Component) on page 925

Adjust command (Component) on page 925

Opposite command (Component) on page 925

Rotate command on page 926

Shove command (Component) on page 927

Swap command on page 928

Lock command (Component) on page 928

Fix command on page 929

Alternate Footprint command (Component) on page 929

Properties command (Component) on page 929

Delete command (Component) on page 929

Select Tool command (Component)

Using the Component Select Tool command (or the component tool on the toolbar), you can perform editing operations on one or more components.

Select Filtered command (Component)

Select Filtered establishes a component filter, for making a set of components available to you in sequence. For example, you may specify that Layout select only those unplaced ICs with 20 or more pins.

When you choose the Select Filtered command, Layout displays the Component Selection Criteria dialog box for category selection. The selected components attach to the pointer, and you can then use Quick Place from the pop-up menu to place all the selected components, or use Select Next to place each of the selected components individually. To see the name of the next component to be placed, use the Place command.

A Component Group can be set on the schematic to be used with Select Filtered in Layout.

Queue for Placement command (Component)

Use Queue for Placement to select categories of components for placement. When you choose the Queue for Placement command, Layout displays the Component Selection Criteria dialog box for category selection. You can then use Select Next to place each of the selected components individually. To see the name of the next component to be placed, use the Place command.

A Component Group can be set on the schematic to be used with Select Filtered in Layout.

Place command (Component)

Use the Place command to view the name of components and place them individually, after selecting them using the Select Filtered, Queue for Placement, or Select Any commands.

New command (Component)

This command displays the Add Component dialog box so you can define and insert a new component.

Copy command (Component)

Copy creates a copy of the selected components and automatically increments the name.

Shortcut

Keyboard: INSERT

If you do not have a component selected, and you choose New, the Add Component dialog box appears, so you can assign a reference designator and footprint to the new component. If the footprint does not yet exist on the board, it is pulled in from the specified library.

Adjust command (Component)

When you are working interactively, the Adjust command optimizes the components you have selected for aesthetic and routing quality, with the surrounding components in the X and Y axis

Shortcuts

Keyboard: CTRL+J

Opposite command (Component)

Use this command to mirror text, components, groups, and obstacles to the opposite layer. See Layers spreadsheet for a description of the opposite layer.

Single selection of text, components, groups, or obstacles

When you choose the Opposite command, a dotted version of the selected text, component, or obstacle appears on your

pointer, mirrored on the X axis, and automatically moved to the opposite layer (according to the mirror layer listed on the Layers spreadsheet).

You can use this command on components whether the Reconn Enabled button of the toolbar is selected or not selected.

Multiple selection of text, components, groups, or obstacles

If you select multiple instances of text, components, groups, or obstacles, and press T for opposite, each instance is mirrored on its own X axis and automatically moved to the opposite layer (according to the mirror layer listed on the Layers spreadsheet).

You can use this command when Reconn Enabled is selected.

Shortcuts

Keyboard: ALT, T, G, T

T

Rotate command

Use this command to rotate components, groups, text, and obstacles. Selected objects rotate according to the increment values specified in the System Settings dialog box. You can rotate objects counterclockwise or clockwise. A positive value creates a counterclockwise rotation, and is the default direction.

Components, groups, and obstacles can be rotated, even when Reconnect mode is on.

Rotating a component, group, obstacle, or text

When you choose the Rotate command, the selected object rotates about the centroid. To continue rotating the object, press R repeatedly.

Rotating multiple components, groups, obstacles, or text

If you select multiple obstacles and rotate them, the relationship between them remains the same. The entire group rotates around the centroid, rather than each object rotating in about its own centroid.

Shortcuts

Keyboard: ALT, T, G, R

R

Shove command (Component)

Layout and Layout Plus only

Use this command to place components in the midst of other components, moving the other components aside.

Shove uses the place outlines from the footprint libraries to determine how closely two components can be placed together. It also pays attention to insertion outlines. The rule is that an insertion outline can overlap another insertion outline, but cannot overlap a place outline.

Shortcuts

Keyboard: J

Swap command

You can swap a component with another component, a pin with another pin, or a gate with another gate.

To swap components, pins, or gates

1. Select the first object you want to swap.
2. From the Tool menu, point to the object type you are swapping (component, pin, or gate), and select Swap.

   If you are swapping gates, Layout highlights the gates that are available for swapping.
3. Select the second object you want to swap.

Note: You should not attempt any gate or pin swaps unless you are sure that you will be able to back annotate the changes to your schematic.

Shortcuts

Keyboard: CTRL+W

Lock command (Component)

You can temporarily lock a component or group in a specific location.

You can unlock a locked object by selecting it and answering Yes when Layout asks if you want to override the lock.

You can also unlock an object by opening the Components data spreadsheet, selecting the component, opening the Edit Component dialog box, and clearing the Fixed check box.

Shortcuts

Keyboard: ALT, T, G, L

L

Fix command

Fix Components permanently fixes the location of selected components and groups.

You can only override the Fix command by opening the Components data spreadsheet, selecting the component, opening the Edit Component dialog box, and clearing the Fixed check box.

Shortcuts

Keyboard: ALT, T, G, F

Alternate Footprint command (Component)

Use the Alternate Footprint command to assign an alternate footprint as defined in the FPLIST attribute. When you choose this command, Layout displays the Footprint Selection dialog box, from which you can choose an alternate footprint.

Properties command (Component)

After selecting a component, choosing the Properties command displays the Edit Component dialog box.

Shortcuts

Keyboard: CTRL+E

Delete command (Component)

The component Delete command deletes a single component or multiple components.

To delete components

1 Single or multiple select the component(s) you want to delete.

2 After the selection, do one of the following:

- Click the Delete button on the toolbar.
- Press the DELETE key on the keyboard.
- From the Edit menu, choose Delete.
- From the Tool menu, point to Component and choose Delete.

Shortcuts

Keyboard: CTRL+X

Gate Flyout Commands

Select Tool command (Gate) on page 930

Select Tool command (Gate)

Use the Select Tool command to perform gate swapping.

To use Select Tool to perform gate swapping

1 Select the first gate you want to swap.

2 From the Tool menu, point to Gate and select Swap.

Layout highlights the gates that are available for swapping.

3 Select the second gate you want to swap.

Shortcut

Keyboard: ALT, T, A

Footprint Flyout Commands

New command (Footprint) on page 931

Delete command (Footprint) on page 932

New command (Footprint)

When the Footprint data spreadsheet is active, you can use the New command to create a new footprint, or add a pad to an existing footprint.

To add a new footprint

1. From the Tool menu, point to Footprint and select the Select from Spreadsheet command.
2. Find and highlight a footprint definition you want to use as a starting point.
3. From the Tool menu, point to Footprint and select New.

   The Add Footprint dialog box appears.
4. Set the new footprint options to the settings you want.
5. Click OK.

Layout adds the new footprint to the bottom of the spreadsheet. Scroll down to view it.

To add a new pad to an existing footprint

1. From the Tool menu, point to Footprint and select the Select from Spreadsheet command.
2. Find the footprint to which you want to add a pad. Highlight the pad definition you want to use as a starting point.
3. From the Tool menu, point to Footprint and select New.

   The Add Pad dialog box appears.
4. Set the new pad options to the settings you want.
5. Click OK.

Layout adds the new pad to the bottom of the footprint definition.

Delete command (Footprint)

The footprint Delete command deletes a single or multiple footprint definition(s).

To delete footprints

1. Single or multiple select the footprint(s) you want to delete.
2. After the selection, do one of the following:

- Click the Delete button on the toolbar.
- Press the DELETE key on the keyboard.
- From the Edit menu, choose Delete.
- From the Tool menu, point to Footprint and choose Delete.

If a footprint that you want to delete is in use in the design, Layout gives you a message that it cannot delete the footprint.

Shortcuts

Keyboard: CTRL+X

Padstack Flyout Commands

New command (Padstack) on page 933

Copy Layer command (Padstack) on page 933

Save to Library command (Padstack) on page 934

Properties command (Padstack) on page 934

Delete command (Padstack) on page 935

New command (Padstack)

Before using the New command to create a new padstack definition, you must activate the padstack data spreadsheet, by using the Select from Spreadsheet command.

To create a new padstack layer

1. From the Tool menu, point to Padstack and select the Select from Spreadsheet command.
2. From the Tool menu, point to Padstack and select New.

   Layout places a new padstack definition at the end of the spreadsheet. Scroll down to view it.
3. From the Tool menu, point to Padstack and select Properties.
4. Name the padstack and set the properties for each layer appropriately.

If you want to, you can save the new padstack definition to your choice of libraries.

To save the new padstack definition to a library

1. Ensure the new padstack definition is highlighted.
2. From the Tool menu, point to Padstack and select Save to Library.
3. Select a library from the Select Library drop-down list, or click the Browse button to browse for a library name. You can also create a new library by clicking the Create New button and defining a path and file name for the new library.

Copy Layer command (Padstack)

This command is available when the Padstack Spreadsheet is active.

Use this command to copy a padstack description from one layer to another or to add padstack clearance to a particular

layer. You might also use this command to establish the padstack size on the solder layer so that you can quickly set up the solder mask.

When you choose this command, Layout displays the Copy Padstack Layer dialog box.

Shortcut

Keyboard: ALT, T, S, C

Save to Library command (Padstack)

This command is available when the Padstacks spreadsheet is active, and operates on the padstacks that are selected in the Padstacks spreadsheet.

This command displays the Save padstacks - Select library dialog box, which you can use to create a new padstack library or browse to an existing padstack library.

If you try to save a padstack to a library and the padstack already exists in the library, Layout displays a dialog box asking if you want to replace the padstack. If you choose the No button, the attempt to save the padstack is cancelled.

Shortcut

Keyboard: ALT, T, S, S

Properties command (Padstack)

After selecting a padstack, choosing the Properties command displays the Edit Padstack dialog box.

Shortcut

Keyboard: CTRL+E

Delete command (Padstack)

The padstack Delete command deletes single or multiple padstack definitions.

To delete padstacks

1. Single or multiple select the padstack(s) you want to delete.
2. After the selection, do one of the following:

- Click the Delete button on the toolbar.
- Press the DELETE key on the keyboard.
- From the Edit menu, choose Delete.
- From the Tool menu, point to Padstack and choose Delete.

Shortcuts

Keyboard: CTRL+X

Pin Flyout Commands

Select Tool command (Pin) on page 936

Copy command (Pin) on page 936

Rotate command (Pin) on page 937

Toggle Copper Pour Seed (Pin) on page 937

Properties command (Pin) on page 937

Delete command (Pin) on page 938

Select Tool command (Pin)

Select Tool command (Pin)

With the Pin tool selected, you can select individual pins.

In the Design window, you can swap pins, or enter the Connection dialog box to change the nets belonging to each pin.

In the Library window, you can move pins, insert pins, delete pins, or enter the Footprint dialog box to edit individual pins.

Before using the Pin commands, ensure that the Allow Editing of Footprints option is enabled in the User Preferences dialog box, under the Options menu.

Shortcut

Keyboard: ALT, T, P

Copy command (Pin)

Before using this command, ensure that the Allow Editing of Footprints option is enabled in the User Preferences dialog box, under the Options menu.

You can copy a pin in your design, using this command.

To copy a pin

1. In the design, select a pin you want to copy.
2. From the Tool menu, point to Pin and select Copy.

   Layout attaches a copy of the pin to the pointer for placement.

Shortcut

Keyboard: ALT, T, P, C

CTRL+C

Rotate command (Pin)

Before using this command, ensure that the Allow Editing of Footprints option is enabled in the User Preferences dialog box, under the Options menu.

You can rotate a pin in your design, using this command.

To rotate a pin

1. In the design, select a pin you want to rotate.
2. From the Tool menu, point to Pin and select Rotate.

   Layout attaches a copy of the pin to the pointer for placement.

Shortcut

Keyboard: ALT, T, P, R

R

Toggle Copper Pour Seed (Pin)

This command toggles the copper pour seed status for a pad that you select. If a pad is designated as a copper pour seed, choosing this command removes that designation. If it is not designated as a copper pour seed, this command assigns that designation.

Properties command (Pin)

After selecting a pin, choosing the Properties command displays the .

Shortcut

Keyboard: CTRL+E

Delete command (Pin)

The pin Delete command deletes a single pin.

To delete a pin

1 Select the pin you want to delete.

2 After the selection, do one of the following:

- Click the Delete button on the toolbar.
- Press the DELETE key on the keyboard.
- From the Edit menu, choose Delete.
- From the Tool menu, point to Padstack and choose Delete.

Shortcuts

Keyboard: CTRL+X

Aperture Flyout Commands

New command (Aperture) on page 938

Properties command (Aperture) on page 938

Delete (Aperture) on page 939

New command (Aperture)

You can add a new aperture to the aperture list with this command. Choosing this command displays the Edit Apertures dialog box.

Properties command (Aperture)

After selecting an aperture from the spreadsheet, choosing the Properties command displays the Edit Apertures dialog box.

Shortcut

Keyboard: CTRL+E

Delete (Aperture)

The pin Delete command deletes a single pin.

To delete an aperture

1. Select the aperture you want to delete, from the spreadsheet.
2. After the selection, do one of the following:

- Click the Delete button on the toolbar.
- Press the DELETE key on the keyboard.
- From the Edit menu, choose Delete.
- From the Tool menu, point to Padstack and choose Delete.

Shortcuts

Keyboard: CTRL+X

Net Flyout Commands

Unroute command (Net) on page 940

Unroute Partial Track (Net) on page 940

Lock command (Net) on page 940

Unlock command (Net) on page 941

Enable <-> Disable command (Net) on page 941

Assign Via per Net command (Net) on page 941

Change Color command (Net) on page 942

Change Width command (Net) on page 942

Force Width by Layer command (Net) on page 942

Force Min/Max Widths command (Net) on page 943

Properties command (Net) on page 943

Delete command (Net) on page 944

Unroute command (Net)

This command unroutes an entire net and allows you to reroute it. It is available when any routing tool is selected in the Tool menu.

Unroute Partial Track (Net)

The command is available when the Nets spreadsheet is active.

Use this command to remove any partially completed connections (those that have no wiring from pad-to-pad). This command removes all the partial tracks of the selected nets. It is most often used when trying to complete the routing on a difficult board.

Shortcut

Keyboard: ALT, T, N, P

Lock command (Net)

Use this command to lock all tracks associated with a net.

Shortcut

Keyboard: ALT, T, N, L

CTRL+L

Unlock command (Net)

Use this command to unlock all tracks associated with a net.

Shortcut

Keyboard: ALT, T, N, U

CTRL+L

Enable <-> Disable command (Net)

This command is available when the Nets spreadsheet is active.

This command is a toggle to enable and disable nets for routing.

The Enable <-> Disable command is especially useful during those times when you want to route only the power and ground nets, then route the signal nets while power and ground are held stable.

Shortcut

Keyboard: ALT, T, N, E

Assign Via per Net command (Net)

This command is available when the Nets spreadsheet is active.

Use this command to assign a specific vias that must be used when routing a particular net. Assigning a via to a particular net does not prohibit any other net from using that via.

When you choose this command, Layout displays the Assign Via dialog box. A via is present in the Assign Via dialog box only after it has been defined in the Edit Padstack dialog box.

Shortcuts

Keyboard: ALT, T, N, V

Change Color command (Net)

This command is available when the Net spreadsheet is active and a net is selected.

You can change the color in which Layout displays specific unrouted nets.

To change the color of an unrouted net

1. From the Tool menu, point to Net and choose the Select from Spreadsheet command.
2. Select the net for which you want to change color.
3. From the Tool menu, point to Net and choose Change Color, or choose Change Color from the pop-up menu.

   A box appears from which you can choose a color.
4. Select the color you want to use.

Change Width command (Net)

Change Width changes the width of a segment you are routing and can only be invoked when you are actively routing with one of the manual routing modes: Shove Track Mode, Edit Segment Mode and Add/Edit Route Mode.

Shortcuts

Keyboard: ALT, T, N, W

W

Force Width by Layer command (Net)

This command is available when the Nets spreadsheet is active.

In the Net Widths by Layer dialog box, you specify the width for a particular net on a specific layer. If you should interactively change the width of the net, you can use this command to force the net to the specified width.

Shortcut

Keyboard: ALT, T, N, F

Force Min/Max Widths command (Net)

This command is available when the Nets spreadsheet is active.

Use this command to force all of the pre-routes on a given net to conform to the range specified by the Min Width and Max Width values. Any tracks wider than the Max Width are forced down to the Max Width value; any tracks narrower than the Min Width are forced up to the Min Width value.

For example, if you have 50 mils routing on the board, and you determine that 25 mils is sufficient, you can change the Max Width of the appropriate nets from 50 to 25, then choose this command to force all of the tracks down to that width.

Shortcut

Keyboard: ALT, T, N, R

Properties command (Net)

After selecting a net from the spreadsheet, choosing the Properties command displays the Edit Net dialog box.

Shortcut

Keyboard: CTRL+E

Delete command (Net)

The net Delete command deletes a single net, or multiple-selected nets.

To delete nets

1. From the spreadsheet, select the net(s) you want to delete.
2. After the selection, do one of the following:
   - ❑ Click the Delete button on the toolbar.
   - ❑ Press the DELETE key on the keyboard.
   - ❑ From the Edit menu, choose Delete.
   - ❑ From the Tool menu, point to Padstack and choose Delete.

Shortcuts

Keyboard: CTRL+X

Connection Flyout Commands

Select Tool command (Connection) on page 944

Tack command (Connection) on page 945

Add command (Connection) on page 945

Delete command (Connection) on page 946

Disconnect Pin command (Connection) on page 946

Select Tool command (Connection)

The connection Select Tool command, and the connection button on the toolbar places Layout in the connection mode.

Shortcut

Keyboard: ALT, T, C

Tack command (Connection)

Use this command to manually tack a connection (place a corner in an unrouted connection as a routing guide) to the outside of the current routing window in such a way that the autorouter finds what you consider to be the best path for the segment. This command is only available when one of the routing tools is selected.

You can place as many corners as you want in a given connection, as long as they are all outside of the routing window. You can only pick up a connection that is at least partially within the routing window.

Shortcut

Keyboard: ALT, T, C, T

CTRL+T

Add command (Connection)

Use this command to add connection to your design.

To add connections

1. From the Tool menu, point to Connection and choose Add, or choose Add from the pop-up menu.
2. Select the pad from which to start the connection.
3. Select the destination pad.

   Layout places a ratsnest to show that the connection has been added.

   Note: Be aware that modifications you make to the netlist with this command are not reflected in the design schematic.

If the pads you select are already part of a signal net, Layout displays a message asking you to confirm the combined nets. Once you confirm the connection, Layout asks you to assign a net name. If the name you assign to the net already exists, Layout asks you to confirm that you want to connect the new net to the net with that name.

Shortcut

Keyboard: ALT, T, C, A

Delete command (Connection)

Use this command to remove a connection from a net.

To remove a connection

1. From the Tool menu, point to Connection and choose Delete, or choose Delete from the pop-up menu.
2. Select the connection that you want to remove.

   Layout asks you to confirm that you want to remove the connection.

Note: Be aware that modifications you make to the netlist with this command are not reflected in the design schematic.

Shortcut

Keyboard: ALT, T, C, D

Disconnect Pin command (Connection)

Use this command to disconnect a pin from a net.

To disconnect a pin

1 From the Tool menu, point to Connection and choose Disconnect Pin, or choose Disconnect Pin from the pop-up menu.

2 Select the pin that you want to disconnect from the net.

Layout asks you to confirm that you want to disconnect the pin.

Note: Be aware that modifications you make to the netlist with this command are not reflected in the design schematic.

Shortcut

Keyboard: ALT, T, C, P

Track Flyout Commands

Select Tool command (Track) on page 947

Mirror command (Track) on page 948

Rotate command (Track) on page 948

Lock command (Track) on page 949

Unlock command (Track) on page 949

Unroute command (Track) on page 950

Select Tool command (Track)

The track Select Tool command places Layout in the Add/Edit Route mode for track editing operations.

Shortcut

Keyboard: ALT, T, T

Mirror command (Track)

Use this command to mirror a track on the X axis. You can then use the Rotate command to orient the track the way you want. You must make a copy of an existing track for this command to be effective.

To mirror a track

1. From the Tool menu, point to Track and choose Select Tool, or choose the Add/Edit Route Mode button.
2. Draw a selection rectangle across the track you want to mirror.
3. From the pop-up menu, choose New, or press the INSERT key.
4. From the pop-up menu, choose Mirror.

Shortcut

Keyboard: ALT, T, T, I

CTRL+M

Rotate command (Track)

Use this command to rotate a track. You must make a copy of an existing track for this command to be effective.

To rotate a track

1. From the Tool menu, point to Track and choose Select Tool, or choose the Add/Edit Route Mode button.
2. Draw a selection rectangle across the track you want to rotate.
3. From the pop-up menu, choose New, or press the INSERT key.
4. From the pop-up menu, choose Rotate.

Shortcut

Keyboard: ALT, T, T, R

R

Lock command (Track)

Use this command to lock a track associated with a connection. You can preroute using pin-outs from SMT or other critical components, lock them down so that the router does not alter them, then run the autorouter to finish each track from the new via or line endpoint to its destination.

If you manually route a locked track, the router informs you that the corner you are trying to alter is locked, and asks if you wish to override. Choose OK to override and manually complete the track or Cancel to drop it.

Shortcut

Keyboard: ALT, T, T, L

L

Unlock command (Track)

This command unlocks the selected track, if it is locked. If the track is not locked, this command has no effect. Unlocking a track allows Layout to move it as needed in order to successfully complete a routing operation for the board representation.

Shortcut

Keyboard: ALT, T, T, U

CTRL+L

Unroute command (Track)

Use this command to unroute an entire track, but leave the connection in place.

To unroute a track

1. From the Tool menu, point to Track and choose Select Tool.
2. Select any portion of the track you want to unroute.
3. From the Tool menu, point to Track and choose Unroute.

Shortcuts

Keyboard: D

Track Segment Flyout Commands

Select Tool command (Track Segment) on page 950

Exchange Ends command (Track Segment) on page 950

Delete command (Track Segment) on page 951

Select Tool command (Track Segment)

The track segment Select Tool command places Layout in the Edit Segment mode for track segment editing operations. You can move a segment, exchange ends, change segment width, and delete.

Shortcut

Keyboard: ALT, T, S

Exchange Ends command (Track Segment)

Use this command while using one of the routing tools for one of the following situations:

- If you are routing ratsnest and you accidentally pick up the wrong end of the connection, you can press x to swap ends without having to release the connection.
- If you are modifying an existing track using Manual Path and you are trying to either split or move a segment and the router is not showing you the best path, press x to view an alternate set of paths.

Shortcut

Keyboard: ALT, T, S, X

X

Change Width command (Track Segment)

Change Width changes the width of a segment you are routing and can only be invoked when you are actively routing with one of the manual routing modes: Shove Track Mode, Edit Segment Mode and Add/Edit Route Mode.

Shortcuts

Keyboard: ALT, T, N, W

W

Delete command (Track Segment)

Use this command to delete the track segment, but leave the connection in place.

To delete a track segment

1. From the Tool menu, point to Track Segment and choose Select Tool, or choose the Edit Segment Mode button.
2. Select the track segment you want to delete.
3. From the Tool menu, point to Track Segment and choose Delete, or press the DELETE key on the keyboard.

4 From the pop-up menu, choose End Command.

Shortcuts

Keyboard: CTRL+X

Jumper Flyout Commands

Select Tool command (Jumper) on page 952

New command (Jumper) on page 952

Convert to Components command on page 952

Properties command (Jumper) on page 954

Delete command (Jumper) on page 954

Delete All command (Jumper) on page 954

Select Tool command (Jumper)

The jumper Select Tool command places Layout in the Jumper mode for performing jumper operations. You must designate a jumper layer to make the jumper commands active.

Shortcut

Keyboard: ALT, T, J

New command (Jumper)

The New command displays the Add Component dialog box so you can place jumper components.

Convert to Components command

Layout and Layout Plus only

Using the Convert to Components command, you can replace pseudo jumpers that the autorouter creates while routing a single-layer board. The pseudo jumpers are replaced with jumper components selected from the footprint library.

The single-layer board router will have already created via patterns based on your jumper criteria, which can include horizontal or vertical jumpers only and the length of the jumpers.

You can assign specific footprints from the Layout jumper library (JUMPER.LLB). You can also create your own by selecting the Jumpers option in the Edit Layer dialog box which is available from the Layers spreadsheet.

Once you have routed the board to completion, you should choose the Convert to Components command to convert the routed via patterns into the actual components you have selected from your library.

The dialog box that appears reflects the rules that the router uses while creating the jumpers and lets you verify the criteria for replacing the jumpers with components before actually doing so.

Error markers are created wherever Layout is unable to find a suitable jumper component for a via pattern that exists on the board, or if component-to-component violations are created in the process of inserting the jumpers.

Layout replaces the padstacks originally assigned to the jumpers with the padstacks assigned to the vias that were used during routing, as long as you use the default jumper padstack name J1.

If you do not want the router to route jumpers under components, include a keepout area on the jumper layer in the footprints that you create for the single-layer boards by duplicating the existing "Place outline" onto the jumper layer and converting the copy into a "Route-via keepout."

Shortcut

Keyboard: ALT, T, J, C

Properties command (Jumper)

After selecting a jumper component, choosing the Properties command displays the Edit Component dialog box.

Shortcuts

Keyboard: CTRL+E

Delete command (Jumper)

The jumper Delete command deletes a single or multiple-selected jumper component(s).

To delete jumpers

1. Single or multiple select the jumper(s) you want to delete.
2. After the selection, do one of the following:
   - Click the Delete button on the toolbar.
   - Press the DELETE key on the keyboard.
   - From the Edit menu, choose Delete.
   - From the Tool menu, point to Jumper and choose Delete.

Shortcuts

Keyboard: CTRL+X

Delete All command (Jumper)

Use this command to delete all jumpers in the design.

Via Flyout Commands

Select Tool command (Via) on page 955

New command (Via) on page 955

Add Via command on page 955

Add Free Via command on page 956

Change Via Type command on page 956

Properties command (Via) on page 957

Modify Padstacks command on page 957

Delete command (Via) on page 957

Select Tool command (Via)

The Select Tool command places Layout in the via mode.

Shortcut

Keyboard: ALT, T, V

New command (Via)

Use this command to define and add a new free via. Layout displays the Add Free Via dialog box so you can define the options for the new free via. After you define the new free via, Layout attaches it to the pointer for placement.

Add Via command

Use this command to automatically clear a space for a via in the location you choose. You can only create a new via when you are actively routing in the Auto Path or Manual Route modes.

If you want to use a via other than the first via listed in the Padstacks (Data Spreadsheets)C0373, choose the Change Via Type command command.

Note: Yellow triangles in the ratsnest indicate unrouted, zero-length connections (connections that lead directly from a pad on the top layer to a pad on the bottom

layer). These connections need to be routed using a via.

Shortcuts

Keyboard: ALT, T, V, V

V

Add Free Via command

A free via is a via that is defined and controlled by the user. The system does not automatically insert or delete free vias. Use this command to automatically clear a space for a free via in the location you choose. You can only add a new free via when you are actively routing in the Auto Path or Manual Route modes.

If you want to use a via other than the first via listed in the Padstacks spreadsheet, choose the Change Via Type command.

Note: Yellow triangles in the ratsnest indicate unrouted, zero-length connections (connections that lead directly from a pad on the top layer to a pad on the bottom layer). These connections need to be routed using a via.

Shortcuts

Keyboard: ALT, T, V, E

E

Change Via Type command

In Layout, the via that you have defined as Via1 is used by default whenever a via is needed. You can use this command to replace Via1 with any other via that is defined in the Padstacks spreadsheet. You can change a via as you route a track, or by selecting the via after you have placed it.

When you choose this command, Layout displays the Via Selection dialog box with a selection of all defined vias.

This command is available when the Tool menu selection is set to Auto Path.

Shortcuts

Keyboard: ALT, T, V, H

Properties command (Via)

After selecting a via, choosing the Properties command displays the Edit Component dialog box.

Shortcuts

Keyboard: CTRL+E

Modify Padstacks command

The Modify Padstacks command displays the Padstack data spreadsheet so you can modify a via definition, or define a via that is undefined.

Delete command (Via)

The via Delete command deletes a single or multiple-selected via component(s).

To delete vias

1. Single or multiple select the via(s) you want to delete.
2. After the selection, do one of the following:
   - ❑ Click the Delete button on the toolbar.
   - ❑ Press the DELETE key on the keyboard.
   - ❑ From the Edit menu, choose Delete.

- From the Tool menu, point to Via and choose Delete.

Shortcuts

Keyboard: CTRL+X

Test Point Flyout Commands

Select Tool command (Test Point) on page 958

New command (Test Point) on page 958

Add Test Point command on page 959

Properties command (Test Point) on page 959

Delete command (Test Point) on page 959

Delete All command (Test Point) on page 960

Select Tool command (Test Point)

The Select Tool command places Layout in the test point mode.

Shortcuts

Keyboard: ALT, T, E

New command (Test Point)

Use this command to define and add a new test point. Layout displays the Add Test Point dialog box so you can define the options for the new test point. After you define the new test point, Layout attaches it to the pointer for placement.

Add Test Point command

The Add Test Point command inserts a via at the location you choose. The command is available only while one of the routing tools is active.

Before you use this command, you must define a via as a test point in the Edit Padstack dialog box.

Shortcuts

Keyboard: P

Properties command (Test Point)

After selecting a test point, choosing the Properties command displays the Edit Component dialog box.

Shortcuts

Keyboard: CTRL+E

Delete command (Test Point)

The test point Delete command deletes a single or multiple-selected test point(s).

To delete test points

1. Single or multiple select the test point(s) you want to delete.
2. After the selection, do one of the following:
   - ❑ Click the Delete button on the toolbar.
   - ❑ Press the DELETE key on the keyboard.
   - ❑ From the Edit menu, choose Delete.
   - ❑ From the Tool menu, point to Test Point and choose Delete.

Shortcuts

Keyboard: CTRL+X

Delete All command (Test Point)

Use this command to delete all test points in the design.

Drill Chart Flyout Commands

Move Drill Chart command (Tool menu) on page 960

Move Drill Chart command (Tool menu)

The Move Drill Chart command moves and redraws the drill chart. The drill chart is visible only if the drill drawing layer is set to visible in the Color spreadsheet.

Shortcut

Keyboard: ALT, T, L

Text Flyout Commands

Select Tool command (Text) on page 961

New command (Text) on page 961

Copy command (Text) on page 961

Mirror command (Text) on page 961

Opposite command on page 962

Rotate command on page 963

Delete command (Text) on page 963

Select Tool command (Text)

With the Text tool, you can create and edit text. When it is active and you have text selected, you can paste the text at another location.

Shortcuts

Keyboard: ALT, T, X

New command (Text)

To create new text, you can:

- point to Text from the Tool menu, and choose New
- select the Text tool and choose New from the pop-up menu.

The Text Edit dialog box appears for you to enter the new text. After typing the new text and closing the dialog box, the new text is attached to the pointer, for placement.

Copy command (Text)

Use the Copy command to copy existing text.

To copy existing text

1. From the Tool menu, point to Text and choose Select Tool, or select the Text tool.
2. Select the text you want to copy.
3. From the pop-up menu, choose Copy. A copy of the text attaches to your pointer.
4. Move the text to the desired location and click the left mouse button to place the text.

Mirror command (Text)

Use this command to mirror obstacles or text on the X axis.

To mirror obstacles or text to the opposite layer, use the Opposite command.

Shortcuts

Keyboard: ALT, T, X, I

CTRL+M

Opposite command

Use this command to mirror text, components, groups, and obstacles to the opposite layer. See Layers spreadsheet for a description of the opposite layer.

Single selection of text, components, groups, or obstacles

When you choose the Opposite command, a dotted version of the selected text, component, or obstacle appears on your pointer, mirrored on the X axis, and automatically moved to the opposite layer (according to the mirror layer listed on the Layers spreadsheet).

You can use this command on components whether the Reconn Enabled button of the toolbar is selected or not selected.

Multiple selection of text, components, groups, or obstacles

If you select multiple instances of text, components, groups, or obstacles, and press T for opposite, each instance is mirrored on its own X axis and automatically moved to the opposite layer (according to the mirror layer listed on the Layers spreadsheet).

You can use this command when Reconn Enabled is selected.

Shortcuts

Keyboard: ALT, T, G, T

T

Rotate command

Use this command to rotate components, groups, text, and obstacles. Selected objects rotate according to the increment values specified in the System Settings dialog box. You can rotate objects counterclockwise or clockwise. A positive value creates a counterclockwise rotation, and is the default direction.

Components, groups, and obstacles can be rotated, even when Reconnect mode is on.

Rotating a component, group, obstacle, or text

When you choose the Rotate command, the selected object rotates about the centroid. To continue rotating the object, press R repeatedly.

Rotating multiple components, groups, obstacles, or text

If you select multiple obstacles and rotate them, the relationship between them remains the same. The entire group rotates around the centroid, rather than each object rotating in about its own centroid.

Shortcuts

Keyboard: ALT, T, G, R

R

Delete command (Text)

The text Delete command deletes a single or multiple-selected text instance(s).

To delete text

1. Single or multiple select the text you want to delete.
2. After the selection, do one of the following:
   - ❑ Click the Delete button on the toolbar.
   - ❑ Press the DELETE key on the keyboard.
   - ❑ From the Edit menu, choose Delete.
   - ❑ From the Tool menu, point to Text and choose Delete.

Shortcuts

Keyboard: CTRL+X

Dimension Flyout Commands

Select Tool command (Dimension) on page 964

New command (Dimension) on page 964

Move Datum command on page 965

Properties command (Dimension) on page 965

Delete command (Dimension) on page 966

Select Tool command (Dimension)

Select Tool places Layout in the Auto Dimension mode.

Shortcut

Keyboard: ALT, T, D

New command (Dimension)

Use the New command to create dimension objects, including arrows, lines, and text, by dragging your pointer from a start point to an end point.

Move Datum command

You can move the design datum (origin 0,0) for easy reference and measurement.

If you are in the design window, you can move the board datum. In the footprint library, you move the footprint datum. Unless the datum is moved, pad 1 is the datum of a footprint.

The location of datum is constrained to the Place grid. You may need to change the grid in order to place the datum at a precise location.

Note: It is suggested that you put your datum in the lower left corner of the board design.

Shortcut

Keyboard: ALT, T, D, M

Properties command (Dimension)

You can create Relative dimensions or Absolute dimensions (as measured from the board origin or datum). You can also choose the size of the lines and text that you want to use, the layer the dimensioning appears on, and the type of arrowhead that you wish to use.

To set dimension properties

1. From the Tool menu, point to Dimension and choose Select Tool.
2. Select a dimension.
3. Click the right mouse button and choose Properties from the pop-up menu.

The Autodimension Options dialog box appears for you to set the properties as you want.

Shortcuts

Keyboard: CTRL+E

Delete command (Dimension)

To delete dimensions

1. From the Tool menu, point to Dimension and choose Select Tool.
2. Select a dimension.
3. Click the right mouse button and choose Delete from the pop-up menu.

Shortcuts

Keyboard: CTRL+X

Measurement Flyout Commands

Select Tool command (Measurement) on page 966

Clear command (Measurement) on page 967

Select Tool command (Measurement)

The Select Tool command places Layout in the Measurement mode. In the Measurement mode, you can find the incremental distance between two points, or the total distance of a series of incremental points, and temporarily display the result in the status bar.

To find the distance between two points

1. From the Tool menu, point to Measurement and choose Select Tool.
2. Click on the first point. Layout displays zero in the DIST: and TOTAL: fields, in the status bar.

3 Click on the second point. Layout draws a line between the two points and displays the distance in the DIST: and TOTAL: fields, in the status bar.

Note: If you do not see a measurement line, it is possible that the Global Layer is invisible. The Measurement tool uses the Global Layer to draw this line. To see the line as you create a measurement, click the Color Settings button on the toolbar, and make the Global Layer visible in the Color Settings spreadsheet.

To find the total distance of a series of incremental points

1 Find the distance between the first two points.

2 Click on the third point. Layout displays the incremental distance between the second and third point in the DIST: field, and the total distance between the first and third points in the TOTAL: field.

3 Repeat step 2 until you have measured all the points you need.

Shortcuts

Keyboard: ALT, T, M

Clear command (Measurement)

Use the Clear command to clear the measurement line from the screen, and the measurement fields in the status bar.

Obstacle Flyout Commands

Select Tool command (Obstacle) on page 968

New command (Obstacle) on page 968

Opposite command on page 969

Rotate command on page 970

Exchange Ends command (Obstacle) on page 971

Finish command on page 971

Delete command (Obstacle) on page 972

Select Tool command (Obstacle)

The Obstacle Tool lets you add, move, or delete entire obstacles, and individual obstacle vertices.

In Layout, obstacles are any electrical or non-electrical object other than a pad, track, or text object. Obstacles can exist in the design window or the footprint library; they may be on a specific layer or all layers.

To select and edit a single vertex or segment of an obstacle, click on the location you want to begin editing.

If you want to edit or copy an entire obstacle without moving it, select the obstacle using SHIFT+ left mouse button.

You can move the obstacle by using the Move On/Off toggle, the ARROW keys or by pressing the left mouse button and dragging across any portion of the obstacle to create a selection area.

Shortcut

Keyboard: ALT, T, B

New command (Obstacle)

Use this command to create a new obstacle.

When you create a footprint, you use this command to generate, among other things, the place outline. If you create an obstacle in the footprint library, Layout generates a copy of the obstacle for each cell in the matrix.

To create a new obstacle

1. From the Tool menu, point to Obstacle and choose New.
2. Click the right mouse button and choose Properties. The Edit Obstacle dialog box.
3. Set the Obstacle Type and Obstacle Layer options, and other options, if appropriate. This setting becomes the new default.
4. Click the left mouse button at the location of the first corner of the obstacle.

   Obstacle behavior during creation depends on whether the obstacle is filled or not. You do not have to "close" a filled obstacle. Layout shows you a closed shape as you are drawing the obstacle.
5. To complete the task, press the ESC key after the obstacle has been fully defined.

To copy an existing obstacle

1. Select an existing obstacle. You can use either the Select area technique or shift and click the left mouse button.
2. From the pop-up menu, choose Copy.

   As soon as you select the command, a version of the existing obstacle appears on your pointer.
3. Release the copy at the new location by pressing the space bar or clicking the left mouse button.

Opposite command

Use this command to mirror text, components, groups, and obstacles to the opposite layer. See Layers spreadsheet for a description of the opposite layer.

Single selection of text, components, groups, or obstacles

When you choose the Opposite command, a dotted version of the selected text, component, or obstacle appears on your pointer, mirrored on the X axis, and automatically moved to the opposite layer (according to the mirror layer listed on the Layers spreadsheet).

You can use this command on components whether the Reconn Enabled button of the toolbar is selected or not selected.

Multiple selection of text, components, groups, or obstacles

If you select multiple instances of text, components, groups, or obstacles, and press T for opposite, each instance is mirrored on its own X axis and automatically moved to the opposite layer (according to the mirror layer listed on the Layers spreadsheet).

You can use this command when Reconn Enabled is selected.

Shortcuts

Keyboard: ALT, T, G, T

T

Rotate command

Use this command to rotate components, groups, text, and obstacles. Selected objects rotate according to the increment values specified in the System Settings dialog box. You can rotate objects counterclockwise or clockwise. A positive value creates a counterclockwise rotation, and is the default direction.

Components, groups, and obstacles can be rotated, even when Reconnect mode is on.

Rotating a component, group, obstacle, or text

When you choose the Rotate command, the selected object rotates about the centroid. To continue rotating the object, press R repeatedly.

Rotating multiple components, groups, obstacles, or text

If you select multiple obstacles and rotate them, the relationship between them remains the same. The entire group rotates around the centroid, rather than each object rotating in about its own centroid.

Shortcuts

Keyboard: ALT, T, G, R

R

Exchange Ends command (Obstacle)

Use this command to focus the pointer on the end opposite the current selection, when a linear obstacle is selected.

Shortcuts

Keyboard: ALT, T, B, X

X

Finish command

Use this command to automatically close the shape of the selected obstacle, automatically complete an unrouted connection or straighten previously routed segment.

This is especially useful when you are creating a copper pour or an anti-copper zone. It is capable of adding one 90 degree corner to the shape that you have initiated.

The router uses the Manual Route/Shove Rules strategy cost settings to select and clear a path for the track.

Shortcut

Keyboard: ALT, T, T, F

CTRL+F

Delete command (Obstacle)

The obstacle Delete command deletes single or multiple-selected obstacles.

After selecting an obstacle, there are several ways to activate the Delete command:

- Click the Delete button on the toolbar.
- From the pop-up menu, choose Delete.
- Press the DELETE key on the keyboard.
- From the Edit menu, choose Delete.
- From the Tool menu, point to Group and choose Delete.

Shortcuts

Keyboard: CTRL+X

Error Flyout Commands

Select Tool command (Error) on page 972

Delete command (Error) on page 973

Select Tool command (Error)

With the error tool, you can query errors to find their location and type.

Shortcuts

Keyboard: ALT, T, R

Delete command (Error)

Delete is available when you have the Error tool selected, and there are errors in the design. You can either point at an error and press the DELETE key, or select a group of errors and delete them.

Use the DELETE key or the DEL button on the toolbar to execute the Delete command.

Shortcut

Keyboard: CTRL+X

Options Menu Commands

System Settings command on page 974

Colors command on page 974

Color Rules command on page 975

Auto Backup command on page 975

Global Spacing command on page 976

Placement Strategy command on page 976

Place Settings command on page 977

Route Strategies Flyout Commands on page 977

Route Settings command on page 980

Fanout Settings command on page 981

Thermal Relief Settings command on page 981

Jumper Settings command on page 981

Free Via Matrix Settings command on page 982

Test Point Settings command on page 982

Components Renaming command on page 982

Gerber Settings command on page 983

Post Process Settings command on page 983

User Preferences command on page 983

System Settings command

Use this command to set values for display units, grids, and workspaces.

Shortcuts

Keyboard: ALT, O, Y

CTRL+G

Colors command

In the Color spreadsheet, you can easily determine the colors or visibility of layer.

To display the Color spreadsheet, choose Colors from the Options menu, or choose the Color icon from the Toolbar.

To change a color or visibility given in the spreadsheet, double-click on the object or layer to display the Color dialog box. Alternately, you can bring up the pop-up menu and select from the standard list of sixteen colors. You can also use the pop-up menu to toggle an object between visible and invisible.

Objects not listed in the Color spreadsheet cannot be edited, but you can easily add them. From the pop-up menu, choose New to display the Add Color Rule dialog box.

The Default color for a layer controls the color of all objects on the layer, unless the color for that object is otherwise specified.

If the color shown is cross-hatched rather than solid, the object or layer is set to invisible.

You can set up your color preferences, then save them as a strategy file so that they are available for future projects.

Shortcuts

Keyboard: ALT, O, C

Color Rules command

Using the Color Rules command, you can add an object type or a layer to the Color spreadsheet. If an object type or layer is not listed in the Color spreadsheet, it will be invisible. Furthermore, the color and visibility of object types or layers not listed in the spreadsheet cannot be edited.

You can add three categories to the Color spreadsheet: a default color for a layer, a color for an object type, or a color for a specific object type on a specific layer.

If you wish to edit the visibility or color of an object on a particular layer, you must select the appropriate option and specify a layer in the edit box.

Shortcuts

Keyboard: ALT, O, R

Auto Backup command

Use this command to make choices about the automatic backup feature of Layout. You can set the time between design backups plus the number of backup files to store, and specify whether you want Layout to create a backup file between routing sweeps.

If you choose to store five backup files, each time the system backs up the design, it saves your file to the name BACKUP1.MAX. The old BACKUP1.MAX is copied to

BACKUP2.MAX; BACKUP2.MAX is copied to BACKUP3.MAX; BACKUP3.MAX is copied to BACKUP4.MAX; and BACKUP4.MAX is copied to BACKUP5.MAX. The old BACKUP5.MAX is discarded.

The information contained in Backup Settings dialog box is not saved with the strategy files, but is preserved in the Layout ASCII output file.

Shortcut

Keyboard: ALT, O, A

Stackup Settings command

The Stackup Settings command displays the Stackup Editor dialog box, using which you can specify the stackup settings for the Layout board.

Shortcut

Keyboard: ALT, O, K

Global Spacing command

Use this command to set the spacing criteria for each layer.

Shortcuts

Keyboard: ALT, O, G

Placement Strategy command

Use this command to characterize Layout's behavior during placement operations. You can view the placement strategy file using the Place Pass spreadsheet. Edit the strategy file values in the Edit Place Pass dialog box.

Shortcuts

Keyboard: ALT, O, P

Place Settings command

The Place Settings command displays the Place Settings dialog box, in which you can choose configuration options for the Quick Place and Matrix Place editing modes.

Shortcuts

Keyboard: ALT, O, L

Route Strategies Flyout Commands

Manual Route Strategy command on page 977

Route Layers command on page 977

Route Sweep command on page 978

Route Pass command on page 980

Manual Route Strategy command

Use this command to set strategies for Auto Path and Shove Route. To display the Manual Route Strategy dialog box, point to the Route Strategies command from the Options menu, and choose Manual Route.

Shortcuts

Keyboard: ALT, O, S

Route Layers command

The Route Layers command displays the Route Layer spreadsheet. From the spreadsheet you can determine whether or not a layer is enabled or disabled for routing, on

each sweep. You can also determine the primary direction (horizontal and vertical preference) of the layer, the layer cost (the relative cost of each layer in design terms of using a particular layer), and the between pins cost (the cost of routing between IC pins) for each layer.

You can edit those values using the Edit Layer Strategy dialog box.

Route Sweep command

The Route Sweep strategy sets the general routing criteria for the design, such as window size and sweep direction. There are seven sweeps, zero through six. Sweep zero is for manual routing operations and automatic routing within a Route/DRC box. Sweeps one through six are for autorouting operations.

A Route Sweep is defined as the movement of an active routing window through an entire PCB to potentially route the PCB to 100%.

You can edit the values in the Sweep Edit dialog box.

Shortcut

Keyboard: SHIFT+W

SWEEP #0 - (Win/Comp): The purpose of Sweep #0 is to set the criteria for Manual Design, or the autorouting of a single window (DRC/Route Box) or component within the window (Component command (Autoroute)). These cost settings are not used when autorouting the board.

SWEEP #1 - (Preliminary Route): The purpose of first Batch Sweep is to route all of the memory and near-memory on the design most efficiently. On SMT boards, you can enable the router to automatically route SMT pads out to a plated through via.

SWEEP #2 - (Maze Route): The Maze Route Sweep is used for routing the majority of the signal tracks on a typical design, using the full shove and retry capabilities of the router and all cost settings. The router finishes each window to as near to

100% as possible (within its try limit) and then moves on to the next window.

SWEEPS #3, #4, and #5 - (Next 1, 2, and 3): The Next Sweeps are used to zero in on any unrouted connections left on the board, centering the routing window on each one in turn. The cost settings are set up to optimize the probability of getting a 100% route.

SWEEP #6 - (Special Options): This Sweep may be set up to route as much of a board as possible without any vias, reduce the number of vias on a design, or eliminate extra corners.

See Load Strategy command (File menu) for information on the specific options associated with each strategy file.

You can use the Sweep Edit dialog box to change the name of each sweep, the size of the active routing area for each sweep, the overlap for the individual routing windows that comprise a sweep, and the primary and secondary directions of the sweep. In addition, you can determine whether the router uses 45 degree or 90 degree angles while routing.

To display the Sweep Edit dialog box

1 Double-click on an item in the spreadsheet.

While the spreadsheet is displayed, there are also editing commands available from the Edit menu, as well as from the pop-up menu.

The amount of diagonal routing (45 degree corners) that the router is allowed to use can be set by selecting Off, On, or Maximize. Sweep direction can be set by selecting Route Next Connection in the Sweep Edit dialog box for a Next sweep, or by selecting the appropriate buttons within the primary and secondary direction fields. The size of the active routing area can be set by typing the values in the "Route Box: X" and "Route Box: Y" boxes. The overlap between routing windows during an automatic sweep can be set by typing the percentage values in the "Overlap % X:" and "Overlap % Y:" boxes.

Route Pass command

The Route Pass command displays the Route Pass spreadsheet. The Route Pass spreadsheet displays the major routing strategies that Layout uses for each routing pass.

Enabling a pass also enables the corresponding sweep. Because the router follows a precise order when routing connections, it generally uses only one pass per sweep. However, it can use up to three passes before moving on to the next window.

You can edit route pass values in the Edit Route Pass dialog box.

Shortcut

Keyboard: SHIFT+E

You can use the Edit Route Pass dialog box to adjust the settings for the relative cost in design terms of adding vias (Via Cost), the cost of retry (Retry Cost), the total effort (Route Limit) that the router puts into each pass, the number of routing attempts per connection, and which routing algorithms are to be used. Settings in the Edit Route Pass dialog box only affect autorouting. For manual routing settings, use the Manual Route Strategy dialog box (obtained by choosing Manual Route from the Options menu in the RouteStrategies flyout).

To display the Edit Route Pass dialog box

1 Double-click on an item in the spreadsheet.

While the spreadsheet is displayed, there are also editing commands available from the Edit menu, as well as from the pop-up menu.

Route Settings command

Use this command to set manual and automatic routing options. Layout displays the Route Settings dialog box.

Shortcuts

Keyboard: ALT, O, O

Fanout Settings command

Use this command to specify how Layout implements the fanout operation. Layout displays the Fanout Settings dialog box.

Shortcuts

Keyboard: ALT, O, F

Thermal Relief Settings command

Use this command to set the sizes for both small and large thermal reliefs. Layout uses small thermal reliefs in all cases unless you specifically choose a large thermal relief. Layout displays the Thermal Reliefs dialog box.

Click the Thermal Relief button to see a graphic representation of a thermal relief and its properties.

{button Thermal Relief,JumpId(layout.hlp>lay_pic,Thermal_Relief_Graphic)}

Shortcuts

Keyboard: ALT, O, T

Jumper Settings command

If you want the autorouter to use jumpers when you are doing one-sided routing, select Jumper Layer as the Layer Type on the Edit Layer dialog box, then choose the Jumper Attributes button to display the Jumper Lengths dialog box.

If the design uses jumpers, you must use a strategy file that allows for jumpers. Two such files are provided with Layout in

the Data directory. They are JUMPER_H.SF (for horizontal jumpers) and JUMPER_V.SF (for vertical jumpers).

Jumpers are considered footprint, and you can find them in JUMPER.LLB in the LIBRARY directory. Each jumper has a specific length; those in JUMPER.LLB range in size from .100 inches to 1.2 inches.

In the Jumper Lengths dialog box, you specify the jumper lengths and the jumper orientations that are allowed on the layer.

Shortcuts

Keyboard: ALT, O, J

Free Via Matrix Settings command

Use this command to specify free via settings. Layout displays the Free Via Settings dialog box.

Shortcuts

Keyboard: ALT, O, V

Test Point Settings command

Use this command to specify setup information for test points. Layout displays the Test Point Settings dialog box.

Shortcuts

Keyboard: ALT, O, N

Components Renaming command

Use this command to establish the component renaming direction strategy. Layout displays the Rename Direction dialog box for selecting one of eight strategies.

After renaming components, you should back annotate the information to the schematic.

Shortcuts

Keyboard: ALT, O, M

Gerber Settings command

Use this command to set Gerber preferences. Layout displays the Gerber Preferences dialog box. If you choose the Save button, the settings in this dialog box are saved independently of your design (in the LAYOUT.INI file), so that they can persist from one Layout session to the next.

Shortcuts

Keyboard: ALT, O, E

Post Process Settings command

Use this command to view and change post processor settings. Layout displays the Post Process spreadsheet.

Shortcuts

Keyboard: ALT, O, I

User Preferences command

The User Preferences command of the Options menu displays the User Preferences dialog box, in which you can specify the state of certain global parameters.

Shortcut

Keyboard: ALT, O, U

Auto Menu Commands

Refresh Flyout Commands on page 984

Design Reuse command on page 986

Place Flyout Commands on page 987

Unplace Flyout Commands on page 990

Fanout Flyout Commands on page 991

Autoroute Flyout Commands on page 993

Autoroute SPECCTRA Flyout Commands on page 996

Unroute Flyout Commands on page 998

Design Rule Check command on page 1000

Delete Violating Tracks Flyout Commands on page 1000

Cleanup Design command on page 1001

Rename Components command on page 1002

Back Annotate command on page 1002

Run Post Processor command on page 1002

Create Reports command on page 1003

Refresh Flyout Commands

All command (Refresh) on page 985

Minimize Connections command on page 985

Repour command on page 986

Calculate Statistics command on page 986

All command (Refresh)

Use this command to perform all of the refresh functions:

- Minimize Connections command
- Repour command
- Calculate Statistics command

Shortcuts

Keyboard: ALT, A, H, L

Minimize Connections command

Use this command to perform a connection-length minimization on nets or ratsnests. When you choose Minimize Connections, selected nets and connections are minimized. If there are no nets or connections selected, minimization is global.

Minimize Connections, as well as the autorouter, follows the reconnection rules established in the Reconnection Type dialog box. The choices are:

- **None -** Prohibits the router from reconnecting any of the point-to-point connections within a given net. This disables reconnect in both the autorouter and the option Edit/Mincon.
- **Vertical -** Causes the router to seek primarily vertical paths for each connection within a net. Used for VCC and GND.
- **Horizontal -** Causes the router to seek primarily horizontal paths for each connection within a net. Used for VCC and GND.
- **Std. Orthogonal -** Causes the router to seek the easiest path between any two points within a net (usually the shortest distance, but with a predisposition for horizontal or vertical tracks where available). This is the default, and should be used for all routing of standard digital signals.

- **High speed -** Used for ECL nets or any critical net that must be connected in a certain predetermined order.

If you are routing a new connection using a manual tool, Minimize Connections reconnects to the pins that is closest to the pointer and is on the same net.

When you are in Minimize Connections mode and the Shove route or Manual route tool is active, Layout continually performs connection length minimization.

Shortcuts

Keyboard: ALT, A, H, M

Design Reuse command

Choose the design reuse command from the Auto menu to place design reuse targets in Layout. To use this feature you must have designed and annotated the source Capture schematic for design reuse.

Repour command

The Refresh button on the toolbar recalculates all of the copper pour on the design based on the new routing and placement configuration.

It is recommended that you refresh the copper pour after you have completed your editing tasks and before you create your final artwork

Shortcuts

Keyboard: ALT, A, H, R

Calculate Statistics command

This command calculates statistics about the number of objects in the current design, rate of completion (for

placement and routing), and board status. You can view the results in the Statistics spreadsheet.

Shortcuts

Keyboard: ALT, A, H, C

Place Flyout Commands

Board command (Place) on page 987

Component command (Place) on page 988

Array command (Place) on page 988

Matrix command (Place) on page 989

Free Via Matrix command (Place) on page 989

Test Points command (Place) on page 989

Board command (Place)

The Board command automatically places all components that are not locked or fixed using the strategy set up in passes 1 - 8 of the Edit Place Pass dialog box.

Assign Clusters determines which sets of components should be placed together, considering their connectivity in relation to other sets of components on the board.

Place Clusters places the assigned clusters in the best location on the board, in relation to the other clusters, and any fixed components.

The Proximity Place algorithm looks at the locations of the clusters as a starting point for the placement of the board, and then runs through thousands of permutations for each cluster,

step by step settling on a placement of the board that optimizes the placement quality.

Swap components takes the placement achieved by proximity, and performs a swap to try to reduce connection length and crossovers. Finally, with all components placed in their approximate optimum positions, Adjust command (Component) takes over to finish the placement, optimizing for aesthetic and routing quality.

Shortcuts

Keyboard: ALT, A, P, B

Component command (Place)

Component automatically places the components you have selected using Pass #0 in Strategy/Place Pass.

To autoplace a group of components, select the components you would like to place and then press P.

Shortcuts

Keyboard: ALT, A, P, C

Array command (Place)

Use this command to place components in a circular pattern on your board. This is especially useful for round test boards.

Shortcuts

Keyboard: ALT, A, P, A

Matrix command (Place)

Use this command to automatically place selected components into an existing matrix.

Shortcuts

Keyboard: ALT, A, P, M

Free Via Matrix command (Place)

Use this command to place a free via matrix in accordance with settings established using the Free Via Settings command.

Shortcuts

Keyboard: ALT, A, P, F

Test Points command (Place)

Use this command to generate test points for your design. Before you can generate test points, you must designate one or more vias as test point vias and specify the nets that are to have test points.

Once these two conditions are met, you can check for the existence of test points by using Design Rules Check and selecting Test Points.

You can generate a list of test points by selecting the Create Reports command from the Auto menu.

Note: In some cases, it is possible to create test points using routing vias, but this is not recommended. If you use routing vias as test points, then try to remove all test points, Layout will try to remove the routing vias that are doing double duty. For test points, you should always choose a via padstack that is not currently used for routing.

Shortcuts

Keyboard: ALT, A, P, T

Unplace Flyout Commands

Board command (Unplace) on page 990

Component command (Unplace) on page 990

Board command (Unplace)

Use this command to remove all components from within the board outline, and have them placed outside the board outline.

Shortcuts

Keyboard: ALT, A, U, B

Component command (Unplace)

Use this command to remove selected components from within the board outline, and have them placed outside the board outline.

Shortcuts

Keyboard: ALT, A, U, C

Fanout Flyout Commands

Board command (Fanout) on page 991

DRC/Route Box command (Fanout) on page 992

Component command (Fanout) on page 992

Board command (Fanout)

Use this command to implement fanout for all SMD components in your board design, according to the settings in the Fanout Settings dialog box.

Note: In previous versions of Layout, fanout capabilities were referred to as "via dispersion." In addition to the change in terminology, the implementation procedure has changed as described in help.

Board fanout offers a higher probability for the router to complete signal routing for dense multilayer SMD designs. Unlike power and ground routing, it is not absolutely necessary to implement fanout for all pads, because the router can usually successfully route those pads for which it could not place a fanout via.

For multilayer SMD boards, the router has a better chance of success if you let the router determine via locations, rather than you manually placing vias. Especially on very dense SMD boards, the sheer number of protected vias that you would have to create could present too many obstacles and block too many channels for the router to work efficiently.

Also, when fine-pitch components are present on the board, it is quite helpful to fan them out first, since this is typically the only way you can disperse all pins without blocking off one or more of the pins in the process.

To run fanout on a single component, choose the Component command (Fanout). Typically, there will only be a few fine pitch components on a board.

After you complete fanout for fine pitch devices, you can select and execute a routing strategy to complete the routing for the board.

Note: In previous versions of Layout, fanout capabilities were referred to as "via dispersion." In addition to the change in terminology, the implementation procedure has changed as described in the online help.

Shortcuts

Keyboard: ALT, A, F, B

DRC/Route Box command (Fanout)

Use this command to implement fanout for all SMD components within a specific window of your board design, according to the settings in the Fanout Settings dialog box.

Note: In previous versions of Layout, fanout capabilities were referred to as "via dispersion." In addition to the change in terminology, the implementation procedure has changed as described in help.

Shortcuts

Keyboard: ALT, A, F, D

Component command (Fanout)

Use this command to implement fanout for a particular SMD component in your board design, according to the settings in the Fanout Settings dialog box. When fine-pitch components are present in the design, you should fan them out first, since

this is typically the only way you can disperse all pins without blocking off one or more of the pins in the process.

You can continue selecting individual components for fanout as long as Layout is in the Fanout mode.

To dismiss the fanout mode, choose another tool or press the ESC key.

Note: In previous versions of Layout, fanout capabilities were referred to as "via dispersion." In addition to the change in terminology, the implementation procedure has changed as described in help.

Shortcuts

Keyboard: ALT, A, F, C

Autoroute Flyout Commands

Board command (Autoroute) on page 993

DRC/Route Box command (Autoroute) on page 994

Component command (Autoroute) on page 995

Halt Autoroute command on page 995

Resume Routing command on page 996

Board command (Autoroute)

Layout and Layout Plus only

This command systematically autoroutes your printed circuit board design from start to finish, automatically sweeping through the design. If you have set any of the fanout commands (Board command (Fanout), Component command (Fanout), or DRC/Route Box command (Fanout)) before you

choose the Autoroute Board command, Layout executes the appropriate fanout operation before any routing sweeps.

You set routing priorities and characteristics using the Route Sweep command, Route Pass command, and Route Layers command.

You can interrupt autorouting at any time by pressing ESC, then restart by choosing the Resume Routing command.

If you want to autoroute a design after you have routed it to completion, you must edit the Route Pass spreadsheet and change the passes from Done to Yes.

Shortcuts

Keyboard: ALT, A, A, B

DRC/Route Box command (Autoroute)

Use this command to route a selected area in the design.

The routing criteria for the route box is set in Sweep #0 --Win/Comp, using the Route Sweep command, Route Pass command, and Route Layers command.

The costs are usually set to be equal to a combination of Sweep #1 -- Preliminary Route, and Sweep #2 -- Maze Route. This lets you use the route box to test a routing strategy as though you are autorouting the entire board.

Note: If you want to use the route box to simulate autorouting the entire design, you can do so by routing each window, then clicking the gray area within the scroll bars (between the arrow and the white box) at the right and bottom of the screen to move to the next location.

Shortcuts

Keyboard: ALT, A, A, D

Component command (Autoroute)

Use this command for routing single complex chips, such as gate arrays and connectors, before you route the entire board, You should create a window (using the Zoom DRC/Route Box command (View menu)) that is only slightly larger than the component. Layout routes the point-to-point connections from the selected component to the edge of the DRC/Route box.

If you have set any of the fanout commands (Board command (Fanout), DRC/Route Box command (Fanout), or Component command (Fanout)) before you begin to route the component, Layout executes the appropriate fanout operation before completing the track.

The routing criteria for routing components is set in Sweep #0 (Win/Comp), using the Route Sweep command, Route Pass command, and Route Layers command.

Shortcuts

Keyboard: ALT, A, A, C

Halt Autoroute command

Use this command, or press ESC, to temporarily halt any of the autorouting commands. When you are ready to start again, use the Resume Routing command.

Shortcuts

Keyboard: ALT, A, A, H

Resume Routing command

Resume Routing can be used whenever you have interrupted an autorouting operation. The router resumes routing from the beginning of the first pass within the current DRC/Route Box. It will not reroute any part of the board that has already been routed during the current sweep.

Shortcuts

Keyboard: ALT, A, A, R

Autoroute SPECCTRA Flyout Commands

Launch SPECCTRA command on page 996

Fanout command on page 997

Route Nets command on page 997

Miter Corner command on page 997

Route Board command on page 997

Setup command on page 998

View Report command on page 998

Launch SPECCTRA command

Use this command to launch SPECCTRA from Layout and interactively route the board in SPECCTRA.

For more information, see Launching SPECCTRA from Layout on page 444.

Shortcuts

Keyboard: ALT, A, T, L

Fanout command

Use this command to fanout specific components in your design using SPECCTRA.

For more information, see Performing fanout using SPECCTRA on page 447.

Shortcuts

Keyboard: ALT, A, T, F

Route Nets command

Use this command to route specific nets in your design using SPECCTRA.

For more information, see Routing specific nets using SPECCTRA on page 448.

Shortcuts

Keyboard: ALT, A, T, R

Miter Corner command

Use this command to miter wire corners in the complete board using SPECCTRA.

For more information, see Mitering wire corners using SPECCTRA on page 449.

Shortcuts

Keyboard: ALT, A, T, M

Route Board command

Use this command to autoroute the board using SPECCTRA.

For more information, see Autorouting the board using SPECCTRA on page 450.

Shortcuts

Keyboard: ALT, A, T, B

Setup command

Use this command to specify the options for autorouting the board using SPECCTRA.

Shortcuts

Keyboard: ALT, A, T, S

View Report command

Use this command to generate placement and routing reports using SPECCTRA.

For more information, see Generating SPECCTRA Reports on page 455.

Shortcuts

Keyboard: ALT, A, T, V

Unroute Flyout Commands

Board command (Unroute) on page 999

DRC/Route Box command (Unroute) on page 999

Net command (Unroute) on page 999

Component command (Unroute) on page 999

Board command (Unroute)

Use this command to unroute an entire board. Layout leaves nets and connections intact.

Shortcuts

Keyboard: ALT, A, O, B

DRC/Route Box command (Unroute)

Use this command to unroute within a specified window. Layout leaves nets and connections intact.

Shortcuts

Keyboard: ALT, A, O, D

Net command (Unroute)

Use this command to unroute all tracks of an entire net. Layout leaves the net and connections intact.

Shortcuts

Keyboard: ALT, A, O, N

Component command (Unroute)

Use this command to unroute a selected component. Layout leaves the connections intact.

Shortcuts

Keyboard: ALT, A, O, C

Design Rule Check command

Layout displays the Check Design Rules dialog box when you choose this command, so you can select design rule options.

Shortcuts

Keyboard: ALT, A, D

Delete Violating Tracks Flyout Commands

Board command (Delete Violating Tracks) on page 1000

DRC/Route Box command (Delete Violating Tracks) on page 1000

Board command (Delete Violating Tracks)

Use this command to have Layout automatically sweep through the entire design, checking for space check errors, and automatically rip up a minimum number of segments to eliminate any electrical short conditions.

This operation lets you accurately judge the impact of engineering change-order changes and provides you with a clean design file for rerouting.

You can also delete violating tracks by selecting the Auto CDE option on the Edit Route Pass dialog box.

Shortcuts

Keyboard: ALT, A, V, B

DRC/Route Box command (Delete Violating Tracks)

Use this command to have Layout automatically sweep through a selected window, checking for space check errors, and automatically rip up a minimum number of segments to eliminate any electrical short conditions.

This operation lets you accurately judge the impact of engineering change-order changes and provides you with a clean design file for rerouting.

Shortcuts

Keyboard: ALT, A, V, D

Cleanup Design command

Use this command to enhance the manufacturability of your board. It automatically smooths, miters, and checks for both aesthetic and manufacturing problems that might have been created in the process of manual or automatic routing. You should cleanup your design at least once, at the end of your design cycle.

Some problems this command corrects include:

- off-grid 90 degree angles
- acute angles
- bad copper share
- pad exits
- overlapping vias.

You can also start the Cleanup Design command by selecting the Auto DFM option on the Edit Route Pass dialog box.

Note: Any special routing of nets that have to be preserved, such as matched length routing, should be locked before cleaning up the design.

Shortcut

Keyboard: ALT, A, L

Rename Components command

Rename Components automatically renames components in the order you specify. To rename components, specify the renaming rule, flag the components you don't want to rename, then use the Rename Components command.

Rename Components uses the following rule when renaming components:

- If all components are renamed, each category of component is renamed from 1 to *n*. If any component retains its name, it becomes the baseline for the rest of the renaming process. For example, if there are four ICs on a board (U1, U2, U3, and U4), and you rename U1, U2, and U3, they are renamed as U5, U6, and U7 because U4 is used as the baseline. If you rename U1, U3, and U4, they are renamed as U3, U4, and U5 because U2 is used as the baseline.

Shortcut

Keyboard: ALT, A, N

Back Annotate command

Use this command to annotate changes in your design, from Layout to Capture.

Shortcut

Keyboard: ALT, A, B

Run Post Processor command

Use this command to create all of the plots that are selected using the Post Process Setup command. The Post Process Setup command displays the Post Process spreadsheet. You can edit the spreadsheet using the Post Processing Setup dialog box.

All Gerber files, DXF files, and other output files designated in the spreadsheet as "Enabled" will be output to disk or directly to the device specified.

Running the post processor creates a special design file (*board_name*.GTD) that is preconfigured for GerbTool. GerbTool reads and writes all standard Gerber formats and PPC-350, features automatic tear-dropping, panelization, venting and thieving, and removal of unused pads and silkscreen on pads (Layout LTD includes Gerber plotting and viewing capabilities only.)

The output board_name.GTD is supplied as input to GerbTool for both regular and extended Gerber.

Shortcut

Keyboard: ALT, A, S

Create Reports command

Use this command to select and generate a variety of Layout reports. Layout displays the Generate Reports dialog box.

Shortcut

Keyboard: ALT, A, R

Window Menu Commands

Cascade command (Window menu) on page 1004

Tile command (Window menu) on page 1004

Arrange Icons command (Window menu) on page 1004

Half Screen command (Window menu) on page 1004

Reset All command (Window menu) on page 1005

1, 2, 3, 4 commands (Window menu) on page 1005

Cascade command (Window menu)

Cascade overlaps all open windows, making it easier to switch between windows.

This command also arranges the icons.

Shortcuts

Keyboard: ALT, W, C

SHIFT+F5

Tile command (Window menu)

Tile Command arranges all windows so that none of them overlap, making it easier to see all windows simultaneously.

This command also arranges the icons.

Shortcuts

Keyboard: ALT, W, T

SHIFT+F4

Arrange Icons command (Window menu)

Arrange Icons arranges all icons along the bottom of the screen.

Shortcut

Keyboard: ALT, W, I

Half Screen command (Window menu)

Half Screen displays Capture alongside Layout. The Half Screen feature is operable only if Capture is running in the background or as an icon.

For the Zoom commands to work properly in half screen mode, you must reset the design window after choosing half screen. To do so, choose the Reset All command from the Window menu.

Shortcut

Keyboard: ALT, W, H

Reset All command (Window menu)

Reset All resets all of the data and graphics windows back to their original state. This is useful if the screen has become cluttered, or if you are finished editing and wish to start the autorouting or autoplacement routines.

Shortcut

Keyboard: ALT, W, R

ALT+R

1, 2, 3, 4 commands (Window menu)

Use the numbers and names listed at the bottom of the Window menu to open one of the Layout windows. Select the number or name that corresponds to the window you want to open.

Help Menu Commands

Layout Help command on page 1006

What's New command on page 1006

Known Problems and Solutions command on page 1006

Web Resources Commands on page 1006

Learning Layout command on page 1007

Manuals command on page 1007

About Layout command on page 1007

Layout Help command

Use this command to display the Help window.

Shortcuts

Keyboard: ALT, H, H

F1

What's New command

Use the What's New command from the Help menu to view the Layout What's New document in the default web browser.

Shortcut

Keyboard: ALT, H, P

Known Problems and Solutions command

Use the Known Problems and Solutions command from the Help menu to view a list of currently known problems and solutions in the default web browser.

Shortcut

Keyboard: ALT, H, K

Web Resources Commands

From the Web Resources menu, you can access a variety of information on the Internet.

Note: You must have a connection to the Internet for these commands to work properly.

Shortcut

Keyboard: ALT, H, W

Learning Layout command

Use this command to run the online, interactive tutorial.

Shortcut

Keyboard: ALT, H, L

Manuals command

Use this command from the Help menu to launch the Cadence Online Documentation System. With this system you can access PDF files of the current user's guides. These formatted files can be easily printed on your office printer.

Shortcut

Keyboard: ALT, H, M

About Layout command

Use this command to get the software version number, licensing information, and registration number.

Shortcut

Keyboard: ALT, H, A

View Spreadsheet Toolbar Button Commands

Strategy command (Spreadsheets) on page 1008

Strategy command (Spreadsheets)

Use the Strategy command to display the strategy spreadsheet windows. Strategy spreadsheet windows contain textual data, similar to that found in a Query window. The difference is that the spreadsheets are laid out so that you can easily edit many items at a time, rather than just one at a time as in a Query window.

You can display a dialog box for editing spreadsheet information by double-clicking on a cell. To select multiple cells, press and hold SHIFT while you select the cells.

Use the strategy spreadsheets to set sweep, pass, layer, spacing, and placement strategies.

To access the strategy spreadsheets

1. From the toolbar, select the View Spreadsheet button.
2. Point to Strategy and select the spreadsheet that you want.
   - Route Pass command (Strategy) on page 1008
   - Route Layer command (Strategy) on page 1009
   - Route Spacing command (Strategy) on page 1010
   - Place Pass command (Strategy) on page 1010

Route Pass command (Strategy)

The Route Pass command displays the Route Pass spreadsheet. The Route Pass spreadsheet displays the major routing strategies that Layout uses for each routing pass.

Enabling a pass also enables the corresponding sweep. Because the router follows a precise order when routing

connections, it generally uses only one pass per sweep. However, it can use up to three passes before moving on to the next window.

You can edit route pass values in the Edit Route Pass dialog box.

Shortcut

Keyboard: SHIFT+E

You can use the Edit Route Pass dialog box to adjust the settings for the relative cost in design terms of adding vias (Via Cost), the cost of retry (Retry Cost), the total effort (Route Limit) that the router puts into each pass, the number of routing attempts per connection, and which routing algorithms are to be used. Settings in the Edit Route Pass dialog box only affect autorouting. For manual routing settings, use the Manual Route Strategy dialog box (obtained by choosing Route Strategies/Manual Route from the Options menu).

To display the Edit Route Pass dialog box

1 Double-click on an item in the spreadsheet.

While the spreadsheet is displayed, there are also editing commands available from the Edit menu, as well as from the pop-up menu.

Route Layer command (Strategy)

The Route Layers command displays the Route Layer spreadsheet. From the spreadsheet you can determine whether or not a layer is enabled or disabled for routing, on each sweep. You can also determine the primary direction (horizontal and vertical preference) of the layer, the layer cost (the relative cost of each layer in design terms of using a particular layer), and the between pins cost (the cost of routing between IC pins) for each layer.

You can edit those values using the Edit Layer Strategy dialog box.

Route Spacing command (Strategy)

The Route Spacing spreadsheet displays the spacing criteria for each layer. Spacing is to the nearest 60th of a mil.

Shortcut

Keyboard: SHIFT+G

To display the Edit Spacing dialog box

- Double-click on an item in the spreadsheet.

While the spreadsheet is displayed, there are also editing commands available from the Edit menu, as well as from the pop-up menu

Place Pass command (Strategy)

Using placement strategy files, you characterize Layout's behavior during placement operations. You can view the placement strategy file in the Place Pass spreadsheet. Edit the strategy file values in the Edit Place Pass dialog box.

Shortcut

Keyboard: SHIFT+A

Pop-up menus

There are a number of pop-up menus that appear in Layout. The specific pop-up menu that appears during the session is a function of the currently selected tool.

Block tool pop-up commands on page 1011

Component tool pop-up menu commands on page 1012

Gate tool pop-up menu commands on page 1013

Pin tool pop-up menu commands on page 1014

Obstacle tool pop-up menu commands on page 1014

Text tool pop-up menu commands on page 1015

Connection tool pop-up menu commands on page 1015

Error tool pop-up menu commands on page 1015

Track tool pop-up menu commands on page 1016

Track Segment tool pop-up menu commands on page 1017

Block tool pop-up commands

End Command (pop-ups)

Select All command (Block)

Minimize Connections command

Undo command (Edit menu)

Options

Select Locked and Fixed

Select from Invisible Layers

Select Entire Track

Pop-up commands while creating a polygonal selection

End Command (pop-ups)

Finish command

Delete command (Block)

Any Angle Corners / 135's and 90's only command (Block tool)

Pop-up commands after a selection is created

End Command (pop-ups)

Rotate command (Block)

Lock command (Block)

Delete command (Block)

Move On/Off command (Block)

Undo command (Edit menu)

Component tool pop-up menu commands

Without component selected

End Command (pop-ups)

New command (Component)

Queue for Placement command (Component)

Select Any command (Edit menu)

Place command (Component)

Select Next command (Edit menu)

Minimize Connections command

Undo command (Edit menu)

With component selected

End Command (pop-ups)

Properties command (Component)

Copy command (Component)

Delete command (Component)

Shove command (Component)

Adjust command (Component)

Matrix command (Place)

Quick Place

Swap command

Rotate command

Opposite command (Component)

Break command (Cluster)

Lock command (Component)

Fix command

Select Next command (Edit menu)

Minimize Connections command

Move On/Off command (Component)

Undo command (Edit menu)

Move On/Off command (Component)

Move On/Off is available from the pop-up menu when you have the Component tool selected. Auto Move lets you to toggle between Move mode and Edit mode in the graphics view.

If you click to select a component, you can immediately begin to move it with the pointer. Select Move On/Off from the pop-up menu to freeze the component in place.

If you select a component using SHIFT+ left mouse button or SHIFT+SPACEBAR, it becomes frozen in place. Select Move On/Off to free the component for moving.

You can select more than one component and select Move On/Off to free them for moving.

There are two additional methods for moving selected components:

1. Press any arrow key to move a grid in the direction of the arrow.
2. Draw another area select box at any location on the board and the component(s) jump to your cursor at that location.

Gate tool pop-up menu commands

Without gate selected

End Command (pop-ups)

Minimize Connections command

Undo command (Edit menu)

With gate selected

End Command (pop-ups)

Swap command

Pin tool pop-up menu commands

Without pin selected	With pin selected
End Command (pop-ups)	End Command (pop-ups)
Minimize Connections command	Properties command (Edit menu)
Undo command (Edit menu)	Swap command
	Toggle Copper Pour Seed (Pin)

Obstacle tool pop-up menu commands

Without obstacle selected	With obstacle selected
End Command (pop-ups)	End Command (pop-ups)
New command (Obstacle)	Properties command (Edit menu)
Any Angle Corners command (Obstacle Tool)	Copy command
Undo command (Edit menu)	Delete command (Obstacle)
	Rotate command
	Mirror
	Opposite command
	Move On/Off
	Undo command (Edit menu)

Any Angle Corners command (Obstacle Tool)

Use this command to toggle between the Any Angle Corners mode, and 135s and 90s only mode. Any Angle Corners specify that line ends are not restricted to 90 and 135 degree angles when you are creating obstacles.

Shortcuts

Keyboard: Y

Text tool pop-up menu commands

Without text selected	With text selected
End Command (pop-ups)	End Command (pop-ups)
New command (Text)	Properties command (Edit menu)
Undo command (Edit menu)	Copy command (Text)
	Delete command (Text)
	Rotate command
	Mirror command (Text)
	Opposite command
	Move On/Off
	Undo command (Edit menu)

Connection tool pop-up menu commands

Without connection selected	With connection selected
End Command (pop-ups)	End Command (pop-ups)
Add command (Connection)	Name Net
Delete command (Connection)	
Disconnect Pin command (Connection)	
Minimize Connections command	
Undo command (Edit menu)	

Error tool pop-up menu commands

Without error selected	With error selected
End Command (pop-ups)	End Command (pop-ups)
Select Next command (Edit menu)	Delete command
Undo command (Edit menu)	

Track tool pop-up menu commands

With track selected

End Command (pop-ups)

Select Next command (Edit menu)

Minimize Connections command

Snap To Grid command (Track)

45 Corners command (Track)

90 Corners command (Track)

Curve Corners command (Track)

Any Angle Corners command (Track)

Undo command (Edit menu)

Without track selected

End Command (pop-ups)

Finish command

Unroute Segment command (Track)

Unroute command (Track)

Unroute command (Net)

Segment command (Track)

Exchange Ends command (Track Segment)

Change Width command (Net)

Add Via command

Add Free Via command

Add Test Point command

Lock command (Track)

Unlock command (Track)

Tack command (Connection)

Change Via Type command

Snap To Grid command (Track)

45 Corners command (Track)

90 Corners command (Track)

Curve Corners command (Track)

Any Angle Corners command (Track)

Track Segment tool pop-up menu commands

Without segment selected

End Command (pop-ups)

Select Next command (Edit menu)

Minimize Connections command

Snap To Grid command (Track)

45 Corners command (Track)

90 Corners command (Track)

Curve Corners command (Track)

Any Angle Corners command (Track)

Undo command (Edit menu)

With segment selected

End Command (pop-ups)

Finish command

Unroute Segment command (Track)

Unroute command (Track)

Unroute command (Net)

Exchange Ends command (Track Segment)

Segment command (Track)

Change Width command (Net)

Add Via command

Add Free Via command

Add Test Point command

Lock command (Track)

Unlock command (Track)

Tack command (Connection)

Minimize Connections command

Snap To Grid command (Track)

This command toggles the Snap to Grid Routing option, in the Route Settings dialog box, between On and Off.

45 Corners command (Track)

The 45 Corners setting allows you to create vertices of 45, 90 and 135 degrees.

90 Corners command (Track)

The 90 Corners setting restricts vertices to 90-degree angles.

Curve Corners command (Track)

The Curve Corners option allows curved vertices.

Any Angle Corners command (Track)

Use this command to create vertices at any angle.

Unroute Segment command (Track)

Use this command to unroute the selected segment of track.

Segment command (Track)

Use this command to change to the Edit Segment mode, to edit the currently selected segment.

New command (Track)

Use this command to create a new duplicate connection.

End Command (pop-ups)

End Command deselects any selected objects, and exits the active command mode.

Glossary

A

active layer — In Layout, the design layer that is currently selected or visible on the screen.

algorithm — A procedure for solving a problem, usually mathematical.

analog circuit — A circuit comprised mostly of discrete components (resistors, capacitors, transistors) that produces data represented by physical variables such as voltage, resistance, or rotation.

annular ring — A circular strip of conductive material that surrounds a hole on the printed circuit board.

ANSI — Acronym for *American National Standards Institute*, an organization formed by industry and the U.S. government to develop trade and communication standards. Internationally, ANSI is the American representative to the ISO (International Standards Organization). See also ASCII.

anti-copper — An area within a fill zone in which copper cannot be placed.

aperture — A hole, similar to the aperture of a camera, that is used for photoplotting. Apertures are available in various sizes and shapes.

aperture list	A text file containing the dimensions for each of the apertures used to photoplot PCB artwork.
arc	A segment defined as an arc (a portion of a circle).
arrow keys	On your computer keyboard, the keys you use to navigate around your screen. Each key is marked with an arrow and is named for the direction in which the arrow points. There is an up arrow, down arrow, left arrow, and right arrow key. Also known as *direction keys*.
ASCII	Acronym for *American Standard Code for Information Interchange*; a seven-bit code---based on the first 128 characters of the ANSI character set---that assigns numeric values to letters of the alphabet, the ten decimal digits, punctuation marks, and other characters such as Backspace, Carriage Return, and Line Feed. ASCII is the most widely used character-coding set, and as such enables different applications and computers to exchange information.
assembly drawing	A document that relates information pertaining to the manufacture of a printed circuit board. This information may include the board outline, component outlines and reference designators, part values, and other documentation.
attribute	See property.
autodimension	In Layout, a tool for measuring and documenting the dimensions of your PCB design.
AutoDFM	Acronym for *automatic design for manufacturability*. Layout checks the board to ensure that it meets manufacturability requirements. This feature automatically smooths, miters, and checks for both aesthetic and manufacturing problems that might have been created in the process of routing.
AutoECO	Acronym for *automatic engineering change order*. AutoECO is a Layout command that enables you to forward annotate a printed circuit board from OrCAD Capture for Windows (or another design capture package) to Layout.

automatic component placement	Software that automatically optimizes component placement on a printed circuit board.
automatic dimensioning	A computer-aided drafting function that automatically generates dimensions, leaders, arrowheads, and other similar items that make up a complete set of documented dimensions.
autorouting	Automatic routing performed by a computer application based on a set of rules called strategies.
axial leads	Leads coming out the ends and along the axis of a resistor, capacitor, or other axial part, rather than out the side.

B

back annotate	In Layout, to send data, such as component renames and gate and pin swaps, back to the schematic.
BCD	Acronym for *binary coded decimal*.
BGA	Acronym for *Ball Grid Array*. Leadless array packaging technology in which solder balls are mounted to the underside of the package and are flowed for attachment to printed circuit boards.
blind via	A via that reaches only one surface layer on one side of a multilayered PCB. See also via.
block	A specific portion of the board design that is marked and manipulated as a single entity.
board template	A file that contains a board outline and some design rules. It may also contain drawing formats, dimensions, preplaced components, and tooling holes. See also technology template.
buried via	A via that does not reach a surface layer on either side of a multilayered PCB. See also via.

C

CAD — Acronym for *computer-aided design*. Software used for general or specialized design uses for architectural, mechanical, or electrical design.

CAE — Acronym for *computer-aided engineering*. Software for analyzing designs created on a computer or elsewhere and entered into the computer. Engineering analysis includes, but it not limited to, structural or electronic circuit analysis.

CAM — Acronym for *computer-aided manufacturing*. Software used in all development phases of an information system including analysis, design and programming.

centroid — Center of mass.

clusters — Components that are grouped according to their interrelationships and placed in close proximity on the board. This keeps the connections on the PCB short so that the board is easier to work with.

COB — Acronym for *chip on board*. Component packaging technology in which bare integrated circuits are attached directly to the surface of a substrate and interconnected to the substrate most often by means of microscopic wires.

component — An element or a part. PCBs are made up of components affixed to a common surface and connected by copper tracks. See also footprint.

component density — The quantity of components on a unit area of a printed circuit board.

component hole — A hole in the printed circuit board that corresponds to a pin or wire of a component. This hole serves the dual function of attaching the component to the board, and establishing the electrical connection between the pin or wire and the remainder of the board circuitry.

component library — A Layout file that contains the footprint patterns for a number of components.

component side — The uppermost layer of a board on which most components are placed. In Layout, the component side of the board is nicknamed TOP and is layer number 1. See also solder side.

component silkscreen — The silkscreen markings of the printed circuit board which appear on the component side. The silkscreen is applied over the solder mask.

component solder mask — The colored, usually translucent coating applied to the board over the etched copper. It protects selected copper from the soldering process.

connection — An unrouted, partially routed, or completely routed path between two pads. In a net with *n* pads, there are exactly *n–1* connections.

copper pour — A method by which a copper zone is filled with a specified pattern, avoiding objects that cross the zone or lie within the zone.

copper zone — An area on a board designed to be covered by a layer of copper when manufactured. Also known as a “metal zone.”

core (dielectric) — Hardened fiberglass- epoxy resin, with copper foil bonded to both sides.

cross hatching — The breaking up of large conductive areas by the use of a pattern of lines and spaces in the conductive material.

cross probing — When intertool communication is enabled in Capture, selecting objects in Capture causes the corresponding objects to be highlighted in Layout. Also, selecting objects in Layout causes the corresponding objects to be highlighted in Capture. Both applications must be open. See also intertool communication.

current layer — See active layer.

D

datum	A specific location (a point) that serves as a reference to locate a printed circuit board pattern or layer for manufacture.
default	A parameter whose value is preset by OrCAD in Layout.
density	On a printed circuit board, the degree to which components are packed on the board. Generally, the density is given as the number of square inches per equivalent IC; a lower number indicates a more dense board.
design rule	A guideline that specifies any of a number of parameters for the printed circuit board. These may include minimum clearance between items that belong to different nets, or connection rules. Also, these rules may include specifications for track width to carry a given current, maximum length for clock lines, termination requirements for signals with fast rise and fall times, and so on.
Design Rule Check (DRC)	A feature that checks the PCB layout for violations of pad and track isolations.
design-rule checking	The use of an algorithm to perform continuity verification of all conductor routing in accordance with appropriate design rules.
DFM	See AutoDFM.
discrete components	Components with three or fewer electrical connections (for example, resistors, or capacitors).
dispersion	See fanout.
DRC	See Design Rules Check.
drill chart	A table that appears on the DRLDWG layer of the board showing the current counts, locations, and sizes of the holes to be drilled into the PCB.
drill diameter	The actual size of the drill body.
DXF	A graphics format created by AutoCAD.

E

ECL — Acronym for *Emitter-Coupled Logic*. A type of bipolar transistor that has extremely fast switching speeds.

EDA — Acronym for *Electronic Design Automation*. Software and hardware tools used to ascertain the viability of an electronic design. These tools perform simulation, synthesis, verification, analysis, and testing of the design.

EDIF — Acronym for *Electronic Design Interchange Format*. A standard published by the EIA (Electronic Industries Association) that defines the semantics and syntax for an interchange format that communicates electronic designs.

editor — A tool used to create or modify a design or text file.

EIC — Acronym for *Equivalent Integrated Circuit*. A standard method for determining the number of components on a board. The EIC is determined by taking the number of component pins on the printed circuit board and dividing by 16.

electrical check — The process of checking the printed circuit board to ensure that the connections thereon match those specified in the netlist.

Extended Gerber — See Gerber (274-X).

F

fanout — The process of creating dispersion vias for SMT devices on your printed circuit board. The dispersion vias are connected to SMT devices by via stringers.

feed-through hole — See via.

fill zone — A zone that defines an area to be filled by copper. See also copper pour.

fine pitch — A class of surface-mount components that is characterized by pins that measure 0.025 inches or less from pad center to pad center.

footprint The physical description of a component. It consists of three elements: padstacks, obstacles, and text.

forward annotation The process of sending netlist data in the form of an .MNL file from Capture (or other schematic capture application) to Layout.

FPGA Acronym for *Field Programmable Gate Array*. A logic chip that is programmable and has a high density of gates.

free via A free via and a normal via differ relatively little: they use the same padstack definition and serve much the same purpose. Unlike a normal via, however, once you specify the location and purpose of a free via it cannot be altered by Layout. Layout regards free vias as stand-alone components: you can shove them, place them in isolation (free of tracks), or connect them to multiple tracks on the same net.

Free vias must have a net attachment, though they do not appear in a schematic or netlist. They are preserved through AutoECO, unless the net or routed track they are connected to is entirely deleted or removed from the board. Free via matrices connected only by unrouted connections are never removed by AutoECO unless the entire net is removed from the board.

You can use free vias for special purposes, such as zero length fanouts of ball-grid array components and the "stitching" of plane layers. Test points can be thought of as free vias that use a test-point padstack.

FTP Acronym for File Transfer Protocol. a highly reliable file transfer protocol that is used almost exclusively over the internet. FTP should be used for both binary and ASCII transfers. However, data files should be transferred in binary format.

G

gate swap The exchange of identically functioning gates in order to decrease route lengths.

Gerber (274-D)	A file format that can be read by Gerber and other photoplotter systems that require separately or previously defined aperture lists.
Gerber (274-X)	A file format that can be read by Gerber photoplotters that accept embedded aperture lists. Also known as Extended Gerber.
Gerber data	The Gerber table is the plot area that will be output to Gerber. This area may include the board and peripheral items such as the drill chart or comments.
Gerber photoplotting	A method of transferring PCB artwork to film.
Gerber table	A method of transferring PCB artwork to film.
global layer	When you load a netlist file, Layout places all connections in the netlist on a global layer.
grid	In Layout, all design elements are placed on the board according to five grids: routing, via, dot, place, and detail.
ground plane	An area on the printed circuit board, usually an entire layer, that provides a common ground connection for all component ground pins and other ground connections.

H

heatsink	A mechanical device that is made of a high thermal-conductivity material that dissipates heat generated by a component or assembly.
heuristics	A method of routing that consists of repeatedly attempting to apply very simple routing patterns to unrouted connections in order to complete the routing quickly and cleanly. Typically, heuristics are used for memory and short point-to-point routing.
highlight	Graphical emphasis that is given to text, components, or other objects when they are selected for an action.
hole	In Layout, the areas where board material must be removed by drilling or milling.

hole legend	See drill chart.
HP-GL	Acronym for Hewlett-Packard Graphics Language, which is a plotter protocol developed by Hewlett-Packard.
HP-GL2	An extension of HP-GL that supports polygon fills, wide lines, and other methods of plotting complex shapes.

I

IC	Acronym for *integrated circuit*.
IGES	Acronym for *initial graphic exchange specification*, which is a graphics format for transferring CAD/CAM information.
interactive routing	Routing in which individual connections are entered into the Layout design file manually by the user with the aid of information such as ratsnests, DRC rules, or DFM rules.
intertool communication	A capability that allows OrCAD for Windows tools to share information for display and transfer.
IPC	Acronym for *Institute for Interconnecting and Packaging Electronic Circuits*. An association in the printed circuit board industry that provides standards to enhance commonality of designs.
isolation	The clearance around a pad, track, zone, or via that defines the nearest approach allowed by conductors of another signal set.
ITC	Acronym for intertool communication. See intertool communication. See also cross probing.
jumper wire	A discrete electrical component or wire that is used to make electrical connections between points for which copper etch does not exist due to board density or some other factor.

J

K

keepout	An area fill within which no routing is allowed.

L

land	In Layout, the copper pad needed for a surface mount pin.
layer	One in a series of planes in a PCB design on which tracks are arranged to connect components. Vias connect tracks and zones between layers. Layout allows a maximum of 30 layers, 16 of which can be copper.
layer marker	An object on a board layer that indicates the layer's physical number as counted from the top layer. Used for copper layers only.
LCC	Acronym for *Leaded Chip Carrier*. A chip carrier that is square and contains pin connectors on all four sides. Implementations include the PLCC (plastic LCC) and CLCC (ceramic LCC).
library	In Layout, a collection of footprint samples or templates designed to facilitate PCB design creation.

M

manual routing	Individual connections, in the form of traces, vertices, arcs, etc., which are entered manually into the PCB design.
matrix	In Layout, a tool that creates a structure on the board that can be divided into cells used for the efficient placement of footprints.
MCAD	Acronym for *Mechanical Computer Aided Design*. CAD software specific to mechanical engineering.

mixed component-mounting technology	A component-mounting technology that uses both through-hole and surface-mounting technologies on the same printed circuit board.
MNL	Acronym for *MAX netlist*. This is the netlist format that is accepted by Layout.
mounting hole	A hole used for the mechanical support of a printed board or for the mechanical attachment of components to a printed board.
multilayer board	A printed circuit board that has multiple layers, separated by dielectric material, with connectivity between layers established by vias or through-holes. This term usually refers to a board with more than two layers.

N

net	A logical construct that originates in a schematic and is transferred to a board to describe required electrical connections. The connections may be completed using vias, tracks, or zones. See also track.
netlist	A file that lists the interconnections of a schematic diagram by the names of the signals, modules, and pins connected together on a PCB. The nodes in a circuit.

O

obstacle	An outline that represents an object on the board that must be taken into account during routing, placement, or copper pour.
opaque graphics	In Layout, these are visual representations of objects on your printed circuit board. These objects are represented in different colors if they occupy the same space in the board.

P

pad	On a PCB, a copper shape on one or more layers (there may be a hole and an isolation surrounding the copper)

	used for connecting a component pin to the PCB. The pad indicates where pins of a component are placed.
padstack	A numbered list of via stack descriptions. Each description contains a pad or via definition, including layer, style, drill diameter, size, offset, and solder mask guard width.
pan	In Layout, the screen automatically scrolls to display successive sections of the board when you move the mouse to the screen edge. You can also smoothly pan from one area of the board to another by holding down the C key and moving the mouse.
PCB	An acronym for *printed circuit board*.
PGA	Acronym for *Pin Grid Array*. A chip package with a high density of pins that is used for large amounts of I/O.
pin	The portion of a component to which an electrical connection can be made.
pin swap	The exchange of identically functioning pins in order to decrease route lengths.
pin to pin spacing	The physical spacing between pins on a device.
pitch	In reference to a component, the distance from the center of a pad to the center of an adjacent pad.
placement	The position of components on a PCB. The process of selecting where components will reside on a PCB.
plane layer	A layer of copper that is dedicated to a single net and may have pads and holes that connect to it or pass through it. Typically used for the power and ground layers, CAD tools output the Gerber file for a plane layer in a negative form, meaning that the areas without copper are identified. Regular copper layers are output in positive form, meaning that the areas with copper are identified.
plated through-hole	A through-hole that establishes an electrical connection between layers of a printed circuit board by way of a metal deposition on the inner surface that defines the hole.

polar placement	The process of placing printed circuit board components using polar coordinates referenced from a user-define pole. Typically, this is used for test fixtures.
post processing	A term used to collectively describe the processes performed after the board has been routed in order to produce manufacturing information (silkscreens, reports, drill tapes, assembly drawings, and so on).
power plane	A copper layer dedicated to a signal that is considered to be a power supply. The ground plane is a power plane that supplies the ground potential. These are thick copper layers typically built on thinnest core provide by a PCB fabrication vendor. Thick copper layer reduce resistance, whereas thin core maximizes the capacitance between the planes.
prepeg (dielectric)	A dielectric made of uncured fiberglass- epoxy resin. Prepeg gets cured or hardened when subjected to heat and pressure.
property	A characteristic of an object that can be edited. A property consists of a name and a value. Examples of property names are part value and color. Their respective property values can be something such as capacitor and red.

Q

query	When you select an object on the screen, Layout displays the object attributes for viewing or editing.

R

radial lead	A lead extending out the side of a component, rather than from the end.
ratsnest	A straight-line connection between two or more pads that indicates an electrical connection in the netlist. Ratsnests serve as a reminder that the pads must be connected, and that, currently, there is no track on the board to make that connection.

reference designator A character string denoting the type of component and a number that is specific to that component.

registration The alignment of a pad on one side of the printed circuit board (or layers of a multilayer board) to its corresponding pad on the opposite side.

routing Placing conductive interconnects between components on a PCB layout. The process of turning nets into tracks.

S

scale To enlarge or reduce a printed circuit board representation when printing or plotting.

schematic A graphical description of an electrical circuit.

screen coordinates The X and Y coordinates reporting the location of the cursor on the screen.

segment The partial track that exists between two adjacent vertices or between a vertex and a pin. (Sometimes the track between two pins is called a segment, although it is usually called a connection.)

signal An electrical impulse of a predetermined voltage, current, polarity, and pulse width.

silkscreen Text or outlines (in ink) on the solder mask, on the top, and sometimes on the bottom of modules for a board. Used for component and identification placement on a PCB.

SMD Acronym for *surface mounted device*. A component that is mounted on a surface layer of a printed circuit board, without penetrating the board. See also surface mount.

SMT Acronym for *surface mount technology*. Printed circuit board technology in which the leads on the chips and components are soldered on top of the board as opposed to through it. The use of SMT results in smaller and faster printed circuit boards. See also surface mount.

solder mask	A negative plot of pads with a guard band around the pads. Also, a lacquer applied to prevent solder from adhering to unwanted areas on the PCB.
solder paste	A solder deposit is applied as a paste used for fixing a component to the board.
solder side	The printed circuit board surface opposite the one on which most components are mounted. Also, the bottom layer of the board.
strategy file	A file that contains either placement or routing parameters for a specific type of board with a specific layer structure.
surface mount	A component mounting technology in which holes are not required.

T

Technology template	A file that contains a board outline and the appropriate design rules, drawing formats, dimensions for spacing and grids, preplaced components, and tooling holes for a specific type of board.
test point	A special point of access to an electrical circuit that is used for electrical testing purposes.
thermal relief	A means of connecting a pad to a larger copper area while minimizing the amount of copper available to conduct heat during the soldering process.
thieving	The process of balancing the amount of copper on both sides of a board so that through-hole plating is consistent from top to bottom during board fabrication.
through-hole technology	The process and components associated with producing a printed circuit board that employs traditional through-hole components.
through-hole via	A via that connects the surface layers on a PCB. See also via.
thrucode	See padstack.

tooling hole	A tooling feature in the form of a hole in a printed board of fabrication panel.
trace	See track.
track	The copper trails in the PCB and the onscreen representation of that copper.

U

V

venting patterns	Patterns etched in the board that allow gases formed during fabrication to escape.
vertex	A logical point at which a track is ended and restarted. A vertex is located at each change of direction on the track.
via	A hole connecting layers of a PCB. A through-hole via connects surface layers of a board. On multilayer boards, a via not reaching a surface layer on one side is called a blind via, and a via not reaching a surface layer on either side is called a buried via.
via stack	See padstack.
via stringer	The copper etch that exists between an SMT pad and a corresponding fanout via. See also fanout.

W

X

X axis	The horizontal or left-to-right direction in a two-dimensional system of coordinates. (This axis is perpendicular to the Y axis.)

Y

Y axis	The vertical or bottom-to-top direction in a two-dimensional system of coordinates.

Z

zero-length connection — An unrouted connection between layers where the end points in the connection have the same X- and Y-coordinates. In Layout, these connections are flagged with yellow triangles.

zone — An area on a PCB layer designated as copper or anti-copper. Copper zones may have net names, while anti-copper zones may not. See also copper pour.

zoom — To change the view of a window, making objects appear larger or smaller. When you zoom out, objects are smaller, and you see more of the board. When you zoom in, objects are larger, but you only see a small portion of the board. You can also zoom in or out of a print preview.

Index

.DO file
    customizing, 451
    FANOUTCOMP.DO file, 453
    MITERCORNER.DO file, 454
    ROUTEBOARD.DO file, 454
    ROUTENET.DO file, 454

Numerics

1, 2, 3, 4 commands (Window menu), 1005
1, 2, 3, 4 file commands, 863
3D effects, 93
45 Corners command (Track), 1017
90 Corners command (Track), 1017

A

about layout, 73
About Layout command, 1007
absolute dimension, 505
accelerators, 109
Acrobat PDF
    creating from a catalog, 566
add
    connection, 945
    test point, 959
Add Color Rule dialog box, 607
Add command, 430
Add command (Connection), 945
Add Component dialog box, 608
Add Footprint dialog box, 611
Add Free Via command, 244–245, 389, 417, 956
Add Free Via dialog box, 613
Add Pad dialog box, 615
Add Test Point command, 959
Add Test Point dialog box, 617
Add Via command, 244–245, 389, 417
add/edit route mode, 394, 403
    activating, 56
adding
    matrix lines, 343
    pins, 431
Adjust command, 360
Adjust command (Component), 925
Adjust Comps pass, 353, 356
adjusting components, 360
algorithms, 357
alignment targets
    creating, 261
All command (Refresh), 985
alphabetic pad labels, 662
Alternate Footprint command, 347–348
Alternate Footprint command (Component), 929
alternate footprints, 347–348
annotate

back annotate to Capture, 1002
annotation
forward, 496
anti-copper, 684
Any Angle Corners / 135's and 90's only command (Block tool), 916
Any Angle Corners command (Obstacle), 1014
Any Angle Corners command (Track), 1018
aperture
gerber preferences, 731
new, 938
aperture list, 890
apertures, 885
deleting, 939
edit, 660
editing, 938
apertures spreadsheet, 87
application settings
editing, 880
reloading, 881
arc
drawing, 416
Arc command, 260, 467
arcs, creating, 260, 467
area selection, 61
Arrange Icons command (Window menu), 1004
array alphabet
editing, 662
Array command (Place), 988
ASCII
MIN format, 850
ASCII files, 852
ASCII netlist
viewing, 139
aspect, text characters, 267
Assign Clusters pass, 352, 356
Assign Via dialog box, 619
Assign Via per Net command, 243, 941
attaching, obstacles to components, 256
attempts, 358
attribute data format, 143
attributes
translation from P-CAD, 163
Auto Backup command, 975
auto CDE, 703
auto menu shortcut keys, 114
auto pan, 93
Auto Path, 949, 955, 971
auto path route mode, 409
activating, 56
allow off-grid routing, 409
maximize 45 corners, 409–410
shove components, 409–410
AutoCAD, 209, 855
Autodimension Options dialog box, 620
AutoECO, 33, 495, 870
comparing the connectivity of a Layout board to a schematic netlist, 502
duplicate pin errors, 39
link footprint to component, 752
linking footprints to components, 42
overview, 35
resolving errors, 42
retaining non-electrical parts during, 500
starting a new board, 39
table of ECO types, 37
using design libraries, 40
AutoECO command, 871
AutoECO/Add Only, 870
AutoECO/Add Only command, 877
AutoECO/Add Override, 870
AutoECO/Add Override command, 878
AutoECO/Net Attrs, 870
AutoECO/Net Attrs command, 879
AutoECO/Override All, 870
AutoECO/Override All command, 875
AutoECO/Override Attrs, 870
AutoECO/Override Attrs command, 873
AutoECO/Override Coords, 870
AutoECO/Override Coords command, 874
automatic backup, 975
autoplace
strategy files, 1010
autoroute

board, 993
component, 995
DRC/Route Box, 994
halting, 995
using SPECCTRA, 450
autorouting, 993–996
gridless, 439
autotool select, 59–60, 94

B

Back Annotate command, 494–495, 1002
back annotating, 494
back annotation, 495
backup, 892, 975
Backup command (File menu), 892
Backup Settings dialog box, 622
ball grid array fanout, 390
BGA fanout, 390
block
any angle corners/135's and 90's only, 916
delete, 916
finish, 971
lock, 915
move on/off, 916
rotate, 915
select all, 914
select tool, 914
Block command, 911
Block tool, 67
board
creating without a netlist, 44
delete violating tracks, 1000
dimension, 965
updating to Release 9 format, 217
Board command, 352, 389, 477
Board command (Autoroute), 993
Board command (Delete Violating Tracks), 1000
Board command (Fanout), 991
Board command (Place), 987
Board command (Unplace), 990
Board command (Unroute), 999
board density, 375
board files, 127
closing, 47
opening, 34, 44
saving, 46
board outline, 684
creating, 229
board outlines, 32
creating, 229
verifying before component placement, 324
board templates, 31–32, 127
custom, 226
selecting, 32
boards
adding mounting holes, 237
creating, 225
custom templates, 226
outline, 225, 378
templates, 225
boundaries, 107
Break command, 362
Break command (Cluster), 917
breaking clusters, 362–363
building component clusters, 337

C

CAD tools
IntelliCAD, 867
Cadence web resources, 1006
CadStar CDI file, 191
CadStar import, 173
CadStar INI file, 174–175, 191
CadStar Input/Output files, 174
CadStar Input/Output files (Export), 191
CadStar layer mapping, 175
CadStar layer mapping (Export), 192
CadStar PCB command (Export), 859
CadStar PCB command (Import), 853
calculate statistics, 985
Calculate Statistics command, 986
Capture, 498, 867

creating a new Layout project from a Capture design, 224
Layout macros, 224
mirroring footprints, 341
preparing a design for Layout, 224
Cascade command (Window menu), 1004
catalog
create catalog additional options dialog box, 651
create catalog dialog box, 649
creating, 563
creating an Acrobat PDF, 566
print catalog dialog box, 773
printing, 564
catalog tool, 561
centering footprint insertion origins, 556
Change Color command, 327, 412
Change Color command (Net), 942
Change Via Type command, 418, 956
Change Width command, 415, 942, 951
changing footprints, 347
Check Design Rules dialog box, 623
checking design rules, 473
chip carrier pad array settings, 780
circle
drawing, 416
circular placement, 363, 365
Circular Placement command, 365
Circular Placement dialog box, 626
Circular tab (Pad Array Generator), 630
Cleanup Design command, 482–483, 1001
Cleanup Design dialog box, 632
Clear command (Measurement), 967
clear screen, 904
Clear Screen command (View menu), 904
Clear Selections command (Edit menu), 900
clearance, copper pour, 466
close, 893
Close command (File menu), 893
closing, board files, 47
cluster, 337
cluster ID, 346
clusters, 352, 356–357
breaking, 362
circles, 360
creating, 917
maximum, 358
placement, 362
placing, 360
Coarse command, 903
color
adding a color rule, 103
assigning a color to a layer, 98
assigning a color to an object, 97
changing net colors, 942
database spreadsheet, 88
deleting a color rule, 103
editing a color rule, 103
rules, 102
Color dialog box, 634
Color Rules command, 975
color setup, 885
loading, 888
save as, 891
Color spreadsheet, 96
color-coding nets, 326–327
colors, 607, 891
adding objects, 101
deleting objects or layers, 101
editing, 96
invisible and visible, 909
setting, 96
Colors command, 55, 75, 96, 974
Colors spreadsheet, 974
commands
Add, 430
Add Free Via, 244–245, 389, 417
Add Via, 244–245, 389, 417
Adjust, 360
Alternate Footprint, 347–348
Arc, 260, 467
Assign Via per Net, 243
Back Annotate, 494–495
Board, 352, 389, 477
Break, 362

Change Color, 327, 412
Change Via Type, 418
Change Width, 415
Circular Placement, 365
Cleanup Design, 482–483
Colors, 55, 75, 96
Component, 55, 388
Components Renaming, 494
Connection, 55
Connection Edit, 431
Copy, 256, 339
Database Spreadsheets, 54, 74, 91
Delete, 54, 340, 431
Density Graph, 375
Design, 375
Design Rule Check, 374, 435, 473
Disconnect Pin, 432
DRC/Route Box, 477
Drill Chart Properties, 516
Drill Tape, 517
EnableDisable, 333, 351, 402
End Command, 410, 412
Error, 55
Exchange Ends, 259, 421
Exit, 47
Fanout, 388–389
Fanout Settings, 389
Find/Goto, 54
Finish, 229, 255
Fix, 329
Force Width by Layer, 416
Free Via Matrix, 417
Free Via Matrix Settings, 417
Gerber Settings, 519
Global Spacing, 74
Library Manager, 51, 54
Load, 227, 380–381
Lock, 329, 421
Make, 362
Matrix, 342–343
Matrix Place, 342
Minimize Connections, 339, 358, 411
Mirror, 259
Move Drill Chart, 516
Move On/Off, 340
New, 34, 227, 229, 237, 331, 371, 467
Obstacle, 55
Open, 54, 227
Opposite, 259, 341
Pin, 55
Place, 337, 352, 417
Place Pass, 74
Placement Strategy, 74
pop-up menus, 1010
Post Process, 91
Post Process Settings, 75, 91, 519
Preview, 509
Properties, 54, 91, 229, 242, 248–249, 251, 256, 344, 419, 431, 461, 467, 494, 519
Query Window, 55, 57
Queue For Placement, 335
Quick Place, 363
Refresh, 56
Remove Tack Point, 420
Rename Components, 494
Rotate, 258, 341
Route Settings, 242, 407, 409
Route Strategies, 74
Run Post Processor, 519
Save, 46, 54
Save As, 46, 227–228
Segment, 259
Select Any, 247, 338, 362
Select Layer, 257
Select Next, 335
Shove, 351, 359
Strategy, 74
Swap, 340
System Settings, 233, 235, 258, 326
Tack, 420
Text, 55
Text Editor, 52
Thermal Relief Settings, 458
Toggle Copper Pour Seed, 464
Unlock, 421

Unroute, 413
Unroute Center Partial, 413
Unroute Net, 413
Unroute Partial Track, 413
Unroute Segment, 413
Unroute Unlocked Track, 413
User Preferences, 92, 358–359
VisibleInvisible, 361
Zoom All, 55, 227
Zoom DRC/Route Box, 387
Zoom In, 54
Zoom Out, 55, 254
Comp Attachment dialog box, 635
COMPFIXED, 503
COMPGROUP, 503
COMPKEY, 503
COMPLOC, 503
COMPLOCKED, 503
component
add component dialog box, 608
clusters, 337
copying, 925
linking to footprints, 372
manual placement, 767
mirroring, 258
move on/off, 1013
new, 924
place, 924
placing, 989
queue for placement, 924
report definition, 520
shoving, 927
Component Attachment dialog box, 636
Component command, 55, 388
Component command (Autoroute), 995
Component command (Fanout), 992
Component command (Place), 988
Component command (Unplace), 990
Component command (Unroute), 999
component data format, 145
component group keepin obstacles, 330
component group keepout obstacles, 330
component height keepin obstacles, 330
component height keepout obstacles, 330
component list and netlist report definition, 520
component renaming, 785
Component Selection Criteria dialog box, 637
component tool
pop-up commands, 1012
components, 635–636
adjusting, 353, 356, 360, 925
attaching text to, 268
autoplacing, 352
break clusters, 917
clusters, 917
coordinates, 345
copying, 339
data spreadsheet, 82
deleting, 340, 929
editing, 344, 663
fix, 929
footprint, 345
gate swapping, 357
group number, 346
grouping, 918
grouping for placement, 332
groups, 362
insertion outlines, 359
locking, 920, 928
mirroring, 341, 918, 925, 962, 969
moving, 340
package, 345
pin swapping, 357
place outlines, 359, 361
placing, 818, 899, 924, 929
properties, 929
proximity, 352, 356
reference designator, 344
renaming, 493, 1002
rotating, 919, 926, 963, 970
rotation, 341, 345
shoving, 359
swapping, 340, 353, 357
value, 345

value text, 266
Components Renaming command, 494
Components Renaming command (Strategy), 982
COMPROT, 503
COMPSIDE, 503
connection
adding, 945
delete, 946
select tool, 944
tack, 945
Connection command, 55
Connection Edit command, 431
connection tool
pop-up commands, 1015
connections
minimize, 985
modifying, 755
pins, 946
connections, duplicate, 410
connectivity
validating connectivity using an IPC-D-356 netlist, 478
Connector Stagger X tab (Pad Array Generator), 643
Connector Stagger Y tab (Pad Array Generator), 646
CONNWIDTH, 504
contrast
high contrast, 904
Convert to Components command, 953
Converting pre-Layout v7.10 split planes, 220
coordinate
finding, 720
coordinates, 345
viewing current, 72
copper area, 684
copper free obstacle, 261
copper pour, 684, 740, 840
anti-copper, 462
clearance, 466
copper area, 462
creating, 462
circular, 467
creating split planes, 398
fast fill, 94
free via matrix, 989
hatch pattern, 467
isolating tracks, 466
manually routing connected nets, 384
obstacles, 84
refreshing, 468
repour, 986
seed point, 464
seeding, 466
toggle copper pour seed, 937
using as a shield, 469
using for connectivity, 94
z order, 466
copy
pins, 936
text, 961
Copy command, 256, 339, 896
Copy command (Component), 925
Copy command (Pin), 936
Copy command (Text), 961
Copy Layer command (Padstack), 933
Copy Padstack Layer dialog box, 648
copying
components, 339
matrix, 343
obstacles, 256
to other layers, 257
tracks, 412
copying padstacks into a padstack library, 559
corners
any angle, 1014
cost
pin and layer, 977, 1009
create new footprint
pad array generator, 764
Create New Footprint dialog box, 652
Create Reports command, 520, 1003
creating

arcs, 260
board outlines, 229
circular obstacles, 260
custom templates, 226
DRC box, 387
footprints, 538
obstacles, 254
padstacks, 242
split planes, 398
text, 263
creating a netlist for Layout, 126
creating a track with curved corners, 396
creating alignment targets, 261
creating Gerber files, 478
creating split planes, 398
cross probing, 498
cursor and pointer information, 69
curve
creating a track with curved corners, 396
Curve Corners command (Track), 1018
custom properties text, 266
customer support, 73
web resources, 1006

D

database spreadsheet
apertures, 87
color, 88
components, 82
drills, 86
error markers, 85
footprints, 80
layers, 77
nets, 83
obstacles, 84
packages, 81
padstacks, 78
post process, 89
route sweep, 978
statistics, 76
text, 84
Database Spreadsheets command, 54, 74, 91
Database Spreadsheets command (View menu), 906
datum, 230
moving, 965
Datum command, 230
datum, moving, 230
delete
block, 916
connection, 946
dimension, 966
jumper, 954
matrix, 922
nets, 944
obstacle, 261
test point, 959–960
text, 963
via, 957
Delete All command (Jumper), 954
Delete All command (Test Point), 960
Delete command, 54, 340, 431, 897
Delete command (Aperture), 939
Delete command (Block), 916
Delete command (Component), 929
Delete command (Connection), 946
Delete command (Dimension), 966
Delete command (Error), 973
Delete command (Footprint), 932
Delete command (Groups), 921
Delete command (Jumper), 954
Delete command (Matrix), 922
Delete command (Net), 944
Delete command (Obstacle), 972
Delete command (Padstack), 935
Delete command (Pin), 938
Delete command (Test Point), 959
Delete command (Text), 963
Delete command (Track Segment), 951
Delete command (Via), 957
delete violating tracks
DRC/Route Box, 1000
Delete Violating Tracks command, 477
deleting
components, 340

dimension objects and text, 507
matrix lines, 343
obstacles, 261
pins, 431
text, 268
density graph
coarse command, 903
fine command, 902
medium command, 903
Density Graph command, 375
density graph window, 70
deselecting objects, 61
design
cleanup, 1001
creating, 25
creating without a netlist, 44
merging MAX files into a design, 348
Design command, 375
Design command (View menu), 902
design library
creating, 570
in AutoECO, 570
Design Rule Check command, 374, 435, 473, 1000
design rules
checking, 623
design rules check
resizing the DRC box, 387
design window, 50
opening the library manager, 530
designing a matrix, 921
dialog box
add color rule, 607
add footprint dialog box, 611
add free via dialog box, 613
add pad, 615
add test point, 617
advanced options, 619
assign via, 619
autodimension options, 620
backup settings, 622
check design rules, 623
circular placement, 626
cleanup design, 632
color, 634
comp attachment, 635
component attachment, 636
component selection criteria, 637
copy padstack layer, 648
create new footprint, 652
drill chart properties, 654
drill properties, 655
edit apertures, 660
edit array alphabet, 662
edit component, 663
edit footprint, 665
edit free via, 668
edit layer, 669
edit layer strategy, 675
edit net, 678
edit obstacle, 684
edit padstack, 690
edit padstack layer, 695
edit place pass, 699
edit route pass, 703
edit spacing, 709
edit test point, 711
fanout settings, 712
find and select item, 719
find coordinate or reference designator, 720
footprint selection, 720
free via matrix settings, 721
free via selection criteria, 723
GenCAD to Layout, 724
generate reports, 727
gerber preferences, 731
hatch pattern, 740
IDF to Layout, 738
jumper lengths, 741
layers enabled for routing, 742
Layout MAX to HYP window, 743
Layout to GenCAD, 744
Layout to GenCAM, 745
Layout to IDF, 747
Layout to IPC-356, 746

Layout to ODB++, 749
Layout to SPECCTRA, 751
manual route strategy, 753
modify connections, 755
modify nets, 756
net selection criteria, 756
net spacing by layer, 757
net widths by layer, 758
obstacle selection criteria, 759
package edit, 760
pad array generator, 764
pin attachment, 766
place settings, 767
post process settings, 769
print/plot, 774
properties, 779–780
reconnection type, 783
rename direction, 785
route settings, 789
save footprint as, 812
save padstack - select library, 813
select footprint, 816
select layer, 818
select next, 818
select padstack for pad array generator, 820
select padstacks, 819
solder mask properties, 780
solder paste properties, 779
SPECCTRA to Layout, 825
stackup editor, 826
sweep edit, 829
system settings, 834
test point selection criteria, 836
test point settings, 835
text edit, 837
text selection criteria, 839
thermal relief settings, 840
track width, 841
user preferences, 842
via selection, 847
workspace settings, 847
dimension
absolute, 505
delete, 966
deleting objects and text, 507
new, 964
properties, 965
relative, 505
select tool, 964
direction
renaming components, 982
disabling nets for routing, 333, 402
disconnect
pin, 946
Disconnect Pin command, 432, 946
display unit settings, 974
distance measurement, 966
clear, 967
dogbone technology (fanning out BGAs), 390
drawing arcs and circles, 416
DRC, 623, 829, 1000
resizing the DRC box, 387
DRC box
creating, 387
moving, 387
DRC/Route box
zoom, 909
DRC/Route Box command, 477
DRC/Route Box command (Autoroute), 994
DRC/Route Box command (Delete Violating Tracks), 1000
DRC/Route Box command (Fanout), 992
DRC/Route Box command (Unroute), 999
drill chart, 86
moving, 960
viewing, 513
Drill Chart Properties command, 516
Drill Chart Properties dialog box, 654
drill chart, changing the size of, 516
drill drawing
viewing, 513
drill holes
viewing, 245
Drill Properties dialog box, 655

drill symbol, 655
Drill Tape command, 517
drill tape, generating, 517
drills, 513
    database spreadsheet, 86
Dual/Quad Inline tab (Pad Array Generator), 658
duplicate, 896
duplicate connections, 410
DXF, 209, 211
    creating multi-layer DXF files by exporting, 216
    creating single layer DXF files through the Post Processor, 215
DXF import and export, 209
DXF to Layout command (Import), 855
dynamic reconnect, 251, 382

E

eck, 827
ECO, 752
ECO commands, 869
Edit Apertures dialog box, 660
Edit App Settings command (Tools menu), 880
Edit Array Alphabet dialog box (Pad Array Generator), 662
Edit Component dialog box, 663
Edit Footprint dialog box, 665
Edit Free Via dialog box, 668
Edit Layer dialog box, 669
Edit Layer Strategy dialog box, 675
edit menu shortcut keys, 111
Edit Net dialog box, 678
Edit Obstacle dialog box, 684
Edit Pad dialog box, 687
Edit Padstack dialog box, 690
Edit Padstack Layer dialog box, 695
Edit Place Pass dialog box, 699
Edit Route Pass dialog box, 703
edit segment mode, 394, 404
    activating, 56
Edit Spacing dialog box, 709
Edit Test Point dialog box, 711
editing
    colors, 96, 634
    components, 344
    footprints, 93
    jumpers, 954
    objects, 91
    obstacles, 256
    padstacks, 558
    spreadsheet information, 75
    text, 837
    vias, 957
editing place pass information, 353
editing segments, 406
editing strategy files, 379
EMN, 738
EMN files, 186
EMP files, 186
Enable/Disable command, 333, 351, 402
Enable/Disable command (Nets), 941
enabling
    layers for routing, 248
    nets for routing, 385
    selection filter, 64
end, 900
End command (Edit menu), 900
End command (pop-ups), 1018
End Command command, 410, 412
error
    deleting, 973
    select tool, 972
Error command, 55
error markers
    database spreadsheet, 85
error tool
    pop-up commands, 1015
errors
    good DRC, 475
    spacing, 1000
errors, querying, 474
Exchange Ends command, 259, 421
Exchange Ends command (Obstacle), 971

Exchange Ends command (Track Segment), 950
exchanging ends of obstacles, 259
exit a command mode, 900
Exit command (File menu), 895
exit Layout, 895
exiting Layout, 47
Export command, 856
exporting
  CadStar PCB files, 191–192, 859
  DXF files, 209, 861
  GenCAD files, 862
  GenCAM files, 863
  Hyperlynx files, 862
  IDF files, 862
  IPC-356 files, 862
  Layout to GenCAD, 188, 744
  Layout to GenCAM, 189
  Layout to Hyperlynx, 207, 743
  Layout to IDF, 186, 747
  Layout to IPC-356, 190, 746
  Layout to ODB++, 194, 749
  Layout to PCB 386+, 208
  Layout to SPECCTRA, 187, 751
  MAX ASCII netlists, 858
  MIN files, 857
  MNL files, 139
  multi-layer DXF files, 216
  ODB++ files, 863
  PADS PCB files, 195–196, 859
  P-CAD PCB files, 198–199, 860
  PCB 386+ files, 859
  Protel PCB files, 201–202, 860
  SPECCTRA files, 863
  Tango PCB files, 204–205, 861
exporting files, 856–857
extension records, 890

F

fanout
  automatic, 389
  ball grid array (BGA), 390
  board, 388, 991
  component, 388, 447, 992
  DRC/Route Box, 992
  manual, 389
  power and ground, 392
  using SPECCTRA, 447
Fanout command, 388–389
Fanout Settings command, 389, 981
Fanout Settings dialog box, 712
fast fill copper pour, 94
file format
  MAX ASCII, 141
file menu shortcut keys, 111
files
  board (MAX), 127
  board template (TPL), 127, 225
  custom templates, 226
  Layout, 119
  LAYOUT.INI, 120
  LAYOUT.LOG, 52, 125
  library, 125
  netlist (MNL), 126
  strategy, 132
  SYSTEM.PRT, 122
  technology template (TCH), 127, 225
  USER.PRT, 123
Find and Select Item dialog box, 719
Find Coordinate or Reference Designator dialog box, 720
Find/Goto command, 54
Find/Goto command (Edit menu), 897
finding, 720, 897
  nets, 756
  obstacles, 759
Fine command, 902
Finish command, 229, 255, 971
Fix command, 329, 929
fixed components, 346
  overriding, 329
FMT apertures, 87
FMT files, 119
FOOTPRINT, 503
footprint, 345
  adding a new, 931

alternates, 720
centering insertion origins, 556
create new, 652
deleting, 932
editing, 665
library conversion warning, 752
link footprint to component, 752
linking components to footprints, 372
pad array generator, 764
replacing, 786
save as, 812
select dialog box, 816
updating using AutoECO, 501
footprint editor, 51, 530
creating new footprint in, 538
footprint libraries, 528
managing, 527
footprint name text, 266
Footprint Selection dialog box, 720
footprints, 528, 894
add footprint dialog box, 611
adding pins to, 539
adding to board, 371
alternate, 348
assigning padstacks to footprint pins, 549
attaching obstacles to, 551
changing, 347
creating, 538
obstacles for, 550
database spreadsheet, 80
editing, 687
labeling, 555
linking footprints to components, 42
mirroring in Capture and Layout, 341
origin, 965
pin attachment, 766
select alternate, 929
setting grids for, 537, 548
SYSTEM.PRT, 122
Force Min/Max Widths command, 943
Force Width by Layer command, 416, 942
forced thermal reliefs, 461
forward annotation, 496
FPLIST, 503
free text, 265
free track, 684
free via
add free via dialog box, 613
adding, 956
edit, 668
modify padstack, 957
selection criteria, 723
free via matrix, 417
Free Via Matrix command, 417
Free Via Matrix command (Place), 989
Free Via Matrix Settings command, 417
Free Via Matrix Settings dialog box, 721
Free Via Selection Criteria dialog box, 723
Free Via Settings command, 982
free vias, 242, 244, 389, 417, 419
matrix settings, 721
settings, 982
full screen cursor, 92
Futurenet, 126, 852
Futurenet Netlist command (Import), 852

G

gate
packages, 81
swapping, 930
gate group, 81
gate swapping, 357
gate tool
pop-up commands, 1013
GATEGROUP, 503
gates
swapping, 928
gates, checking before component placement, 327
GenCAD, 744
exporting from Layout, 188
importing to Layout, 182
GenCAD to Layout command, 855
GenCAD to Layout dialog box, 724
GenCAM, 745

exporting from Layout, 189
Generate Reports dialog box, 727
generating test points, 432
Gerber
creating Gerber files, 478
IPC-356 comparison in GerbTool, 479
plot preview, 515
validating connectivity using an IPC-D-356 netlist, 478
Gerber Preferences dialog box, 731
Gerber Settings command, 519, 983
Gerbers, 660
create, 1002
GerbTool, 868
GerbTool apertures, 87
GerbTool command (Tools menu), 868
global action shortcut keys, 109
global route settings, 789
Global Spacing command, 74, 976
good DRC
adding, 475
changing color, 477
deleting, 476
hiding, 626
prevent deleting, 476
goto, 897
reference designator or coordinate, 720
grid array style for pad array generator, 733
Grid Array tab (Pad Array Generator), 733
grid settings, 834, 974
gridded autorouting
interactive routing tools, 439
shove technology, 438
sweep technology, 438
gridless autorouting, 439
SmartRoute, 439
grids
detail, 235
place, 72, 235
routing, 235, 378
setting, 235
for footprint pins, 537, 548
via, 235, 378
visible, 235
ground
enabling for routing, 385
routing, 382
on SMT boards, 382
on through-hole boards, 382
verifying connection to plane layer, 401
ground and power fanout, 392
ground nets, disabling, 351
ground pins, 392
group
pin and gate, 81
properties, 920
group number, 346
grouping components, 918
grouping components for placement, 332
groups, 362
deleting, 921
fix, 929
locking, 920, 928
mirroring, 918, 925, 962, 969
placing components in, 338
rotating, 919, 926, 963, 970

H

half screen, 1004
Half Screen command (Window menu), 1004
Halt Autoroute command, 995
hatch pattern, 467
Hatch Pattern dialog box, 740
height
text, 267
help, 73
help topics, 1006
help, online, 73
heuristics, 703
hiding routes and connections, 358
High Contrast command (View menu), 904
high speed reconnection type, 250
HIGHLIGHT, 504
highlighting, 498
highlighting nets, 326

hints
routing, 428
hole legend, 655
holes
viewing drill holes, 245
hollow pads, 93
horizontal reconnection type, 250
hotkeys, 109, 111–112, 114–116
HPGL, 89
HYP files, 207
Hyperlynx
exporting from Layout, 207

I

icons
arrange icons, 1004
IDF, 738, 747
IDF files, 165
IDF to Layout command (Import), 855
IDF to Layout dialog box, 738
Import command, 849
importing
CadStar files, 173
CadStar PCB files, 174, 853
DXF files, 209, 855
files, 849
Futurenet Netlists, 852
GenCAD files, 855
GenCAD to Layout, 182, 724
IDF files, 855
IDF to Layout, 165, 738
MAX ASCII Netlists, 852
MIN interchange, 850
MNL files, 139
PADS files, 154
PADS PCB files, 154–155, 853
P-CAD PCB files, 158–159, 161, 854
P-CAD user-defined attributes, 163
PCB 386 files, 183
PCB 386 libraries, 184
PCB 386+ designs, 853
PCB II Netlists, 852
Protel 99SE files, 170
Protel 99SE PCB files, 170–171, 854
Protel PCB files, 167, 854
SPECCTRA files, 855
SPECCTRA to Layout, 181, 825
Tango files, 177
Tango PCB files, 178, 854
improrting
Protel files, 166
inch
measurements in Layout, 231
insertion origin
centering, 555–556
moving, 555
insertion outline, 684
insertion outlines, 359
verifying before component placement, 324
installation setup, 49, 120, 122
IntelliCAD command, 867
interactive route settings, 789
interactive routing, 406
interactive routing tools
auto path route, 439
defined, 439
shove track, 439
intertool communication, 224, 498
intertool communication, in design flow, 29
invisible, making layers, 99
IPC-356
creating, 479
exporting from Layout, 190
Gerber comparison in GerbTool, 479
validating Gerbers using, 478
isolating tracks, copper pour, 466
isolation, 469
ITC, 498
iterations, 357

J

JEDEC alphabetic pad labels, 662
jumper
convert to components, 953
delete, 954

new, 952
properties, 954
select tool, 952
Jumper Lengths dialog box, 741
Jumper Settings command, 981

K

keepin obstacles, 330
keepins, 684
keepout obstacles, 330
keepouts, 684
key components, 347
keyboard shortcuts, 109, 111–112, 114–116
known good DRC, 475
known problems and solutions, 73
Known Problems and Solutions command, 1006

L

layer, 818
assigning a color to a layer, 98
changing the layer of a routed net, 396
color rules, 102
visibility, 99
layer "0", 104
layer cost, 977, 1009
layer mapping
CadStar, 175
layer mapping for DXF translation, 211
layer number shortcuts, 116
layer stack, defining, 238
layer strategy
editing, 675
layers, 104
changing visibility, 99
database spreadsheet, 77
defining layer stack, 238
displaying text on, 268
drop-down list on toolbar, 72
editing, 911
editing nicknames, 669
enabling for routing, 248
library, 326
mirror, 326
net spacing on, 251
preview for post processing, 509
select, 911
setting net width, 248–249
unroute, 911
Layers Enabled for Routing dialog box, 742
Layout design flow, 25
Layout files, 119
Layout help, 73
Layout Help command, 1006
Layout MAX to HYP window, 743
Layout to DXF command (Export), 861
Layout to GenCAD command (Export), 862
Layout to GenCAD dialog box, 744
Layout to GenCAM command (Export), 863
Layout to GenCAM dialog box, 745
Layout to Hyperlynx command (Export), 862
Layout to IDF command (Export), 862
Layout to IDF dialog box, 747
Layout to IPC-356 command (Export), 862
Layout to IPC-356 dialog box, 746
Layout to ODB++ command (Export), 863
Layout to ODB++ dialog box, 749
Layout to SPECCTRA command (Export), 863
Layout to SPECCTRA dialog box, 751
Layout, exiting, 47
LAYOUT.CTL, 119
LAYOUT.INI, 119–120
LAYOUT.LOG, 52, 125
learning layout, 73
Learning Layout command, 1007
Learning Layout tutorial, 73
legend, 655
libraries, 894
copying padstacks into a padstack library, 559
custom, 528
files, 125

library manager, 528
making available to Layout, 531
making unavailable to Layout, 531
managing, 527
using the library manager, 529
library
creating a catalog, 561
creating a PDF, 566
installing a PCB 386 library, 184
updating boards to Release 9 format, 217
Library Conversion Warning dialog box, 752
library layers, 326
library manager, 51, 528–529, 865, 894
footprint editor, 530
pad array generator, 764
starting, 530
Library Manager command, 51, 54
Library Manager command (File menu), 894
Library Manager command (LSession Tools Menu), 865
limits, 107
Link Footprint to Component dialog box, 752
linking components to footprints, 372
LIS files, 119
LLB files, 119
load
strategy files, 886
Load Color Setup command, 888
Load command, 227, 380–381
Load command (File menu), 885
Load Merge board command, 887
Load Post Processing Setup command, 888
Load Strategy command, 886
Load Template File command, 886
loading
placement strategy file, 331
routing strategy file, 380
location, text, 267
lock
block, 915
Lock command, 329, 421, 920, 928
Lock command (Block), 915
Lock command (Net), 940
Lock command (Track), 949
locked components, 346
overriding, 329
lsession, 49
LSESSION.INI, 119

M

macro
Capture to Layout, 224
Make command, 362
Make command (Cluster), 917
manual route, 789, 947
Manual Route Strategy command, 977
Manual Route Strategy dialog box, 753
manual routing, 384, 394
45 corners, 402
90 corners, 402
adding a via matrix, 417
adding tacks, 420
adding vias, 417
any angle corner, 402
changing net colors, 412
changing track width, 415
copying tracks, 412
creating duplicate connections, 410
curve corners, 402
editing free vias, 419
editing vias, 418
Exchange Ends command, 421
forcing a net width on a layer, 416
locking tracks, 421
minimizing connections, 411
removing tacks, 420
routing to an off-grid pad, 421
snap to grid routing, 402
teeing into or out of an existing track, 425
unlocking tracks, 421
Unroute Center Partial command, 413

Unroute command, 413
Unroute Net command, 413
Unroute Partial Track command, 413
Unroute Segment command, 413
Unroute Unlocked Track command, 413
use all via types, 402
manuals, 73
Manuals command, 1007
manufacturability, 703
cleanup design, 1001
manufacturability, ensuring, 473
mapping
CadStar layers, 192
DXF files, 209
PADS layers, 196
P-CAD layers, 199
Protel layers, 202
Tango layers, 178
matrix
adding matrix lines, 343
copying, 343
deleting, 922
deleting matrix lines, 343
designing, 921
free via settings, 721
moving, 343
moving matrix lines, 343
placing, 977
placing components, 342
using free via, 417
Matrix command, 342–343
Matrix command (Place), 989
Matrix Place command, 342
MAX ASCII, 126
MAX ASCII file format, 141
MAX ASCII files, 139
MAX ASCII Netlist command (Export), 858
MAX ASCII Netlist command (Import), 852
MAX files, 119, 127
merging into a design, 348
maximum clusters, 358
MAXWIDTH, 504
maze, 703
measurement
clearing, 967
select tool, 966
measurement, units of, 231
measurements in Layout, 231
Medium command, 903
menu
pop-up, 1010
merging
DXF files into Layout, 209
merging .MAX files, 887
merging MAX files into a design, 348
metric
measurements in Layout, 231
micro BGA, 390
MIN Interchange command (Export), 857
MIN interchange command (Import), 850
minimize connections, 985
Minimize Connections command, 339, 358, 411, 985
minimizing connections, 339
MINWIDTH, 504
mirror
layers, 77
text, 961
Mirror command, 259
Mirror command (Text), 961
Mirror command (Track), 948
MIRRORFOOTPRINT, 503
mirroring, 258
components, 341
footprints in Capture and Layout, 341
layers, 326
obstacles, 259
text, 267
mirroring objects, 258
miter wire corners
using SPECCTRA, 449
MNL files, 119, 126
Modify Connections dialog box, 755
Modify Nets dialog box, 756
Modify Padstacks command, 957
Moire, 261

mounting holes, adding to board, 237
mouse information, 68
move
  block, 916
Move Datum command, 965
Move Drill Chart command, 516
Move Drill Chart command (Tool menu), 960
Move On/Off command, 340
Move On/Off command (Block), 916
Move On/Off command (Component), 1013
moving
  components, 340
  DRC box, 387
  matrix, 343
    lines, 343
  obstacles, 257
    to other layers, 257
  text, 268
moving segments, 406
moving, datum, 230

N

net
  changing the layer of a routed net, 396
Net command (Unroute), 999
net data format, 148
net properties, editing, 247
Net Selection Criteria dialog box, 756
Net Spacing by Layer dialog box, 757
Net Widths by Layer dialog box, 758
NETGROUP, 504
netlist, 224
  comparison in GerbTool, 479
  creating a board without, 44
  creating a netlist for Layout, 126
  report definition, 520
netlists, 32, 126
  contents, 32
  viewing a Layout netlist, 139
nets
  assign via per net, 941
  assigning vias to, 243–244
  change width, 942
  changing net colors, 412
  chaning the track width, 397
  color-coding, 326
  colors, 942
  database spreadsheet, 83
  delete, 944
  disabling, 333, 385, 402
  disconnecting pins from, 432
  editing, 678
  enabling, 385
  enabling or disabling for routing, 941
  force minimum and maximum track width, 943
  force width by layer, 942
  highlighting, 326
  locking, 920, 928, 940
  making invisible, 428
  manual routing, 394
  manually routing nets connected to planes and copper pours, 384
  modifying, 756
  properties, 943
  reconnecting, 783
  routing, 742
  spacing by layer, 251
  splitting, 431
  unlocking, 941
  unroute partial track, 940
  unrouting, 940
  verifying connection to plane layer, 401
  weighting, 326
  width by layer, 248–249
nets, disabling power and ground, 351
NETWEIGHT, 504
new
  dimension, 964
  jumper, 952
  test point, 958
  text, 961
  via, 955
New command, 34, 227, 229, 237, 331, 371, 467

New command (Aperture), 938
New command (Component), 924
New command (Dimension), 964
New command (File menu), 884
New command (Footprint), 931
New command (GerbTool command), 869
New command (Group), 918
New command (Jumper), 952
New command (Padstacks), 933
New command (Test Point), 958
New command (Text), 961
New command (Track), 1018
New command (Via), 955
nickname
    layer, 77
nicknames
    layer, 669
no dynamic reconnect reconnection type, 251, 382
no extension records, 890
none reconnection type, 249
non-electrical parts
    retaining during AutoECO, 500
non-renamed components, 347
Not in Netlist components, 346
not in netlist flag, 500
notes
    product, 73
    product notes, 1006
numberical boundaries for Layout, 107

O

object, 258
    assigning a color to an object, 97
    visibility, 99
obstacle
    any angle corners, 1014
    board outline, 229
    copper free obstacle, 261
    database spreadsheet, 84
    deleting, 261, 972
    exchange ends, 971
    finish, 971
    mirroring, 258
    select tool, 968
Obstacle command, 55
Obstacle Selection Criteria dialog box, 759
obstacle tool
    pop-up commands, 1014
obstacles, 636
    attaching
        footprint, 551
        pins, 551
    circular, 467
    component group keepin obstacles, 330
    component group keepout, 330
    component height keepin, 330
    component height keepout, 330
    copying, 256
    creating, 254
        for footprints, 550
    deleting, 261
    editing, 256, 684
    mirroring, 918, 925, 962, 969
    moving, 257
    rotating, 258, 919, 926, 963, 970
    selecting, 255
        segments, 256
ODB++, 749
    exporting from Layout, 194
off-grid pad, 421
online DRC, 93
    activating, 56
online help, 73, 1006–1007
online tutorial, 73
opaque graphics, 93
open
    1, 2, 3, 4 file commands, 863
Open command, 54, 227
Open command (File menu), 885
Open command (Tools menu, GerbTool command), 869
opening, board files, 34, 44
Opposite command, 259, 341, 918, 925, 962, 969
optimization, 632

options
autodimension, 620
options menu shortcut keys, 114
OrCAD, 867
OrCAD Capture command (Tools menu), 867
OrCAD MIN ASCII file, 857
origin
centering insertion origins, 556
outline, 684
board, 229
outlines
insertion, 359
place, 359, 361
output, 89, 727
post process settings, 769

P

package, 345
name text, 266
package data format, 151
Package Edit dialog box, 760
packages
database spreadsheet, 81
pin swapping, 436–437
pad
add pad dialog box, 615
routing to an off-grid pad, 421
Pad Array Generator, 538–548
Select Padstack dialog box, 541
Settings, 542–548
pad array generator
circular tab, 630
connector stagger x tab, 643
connector stagger y tab, 646
dual/quad inline tab, 658
edit array alphabet, 662
grid array tab, 733
QFP/Chip Carrier tab, 780
Pad Array Generator dialog box, 764
pad array style
circular, 630
connector stagger x, 643
connector stagger y style, 646
dual/quad inline, 658
grid array, 733
QFP/Chip Carrier, 780
pad to pad spacing, 709
pads
editing, 687
PADS ASCII file, 154–155, 195
PADS Import, 154
PADS INI file, 154–155, 195
PADS Input/Output files (Export), 195
PADS layer mapping, 196
PADS PCB command (Export), 196, 859
PADS PCB command (Import), 853
PADS PCB Input/Output files, 154
PADS PCB layer mapping, 155
padstack
copy layer, 933
deleting, 935
editing, 934
modify, 957
save, 813
save to library, 934
selecting for pad array generator, 820
padstack layer
copying, 648
padstacks
assigning to footprint pins, 549
copying padstacks into a padstack library, 559
creating, 242
creating new, 933
editing, 558, 690, 695
saving to a library, 933
selecting, 819
padstacks database spreadsheet, 78
Padstacks spreadsheet, 558–559
panelize, 599
panning, 907
PARTNUM, 503
PARTSHAPE, 504
Paste command, 897
P-CAD INI file, 198

P-CAD Input/Output files, 158
P-CAD layer mapping (Export), 199
P-CAD Options/Defaults, 159
P-CAD PCB command (Export), 198, 860
P-CAD PCB command (Import), 854
P-CAD PCB layer mapping (Import), 161
P-CAD PDIF file, 198
P-CAD PDIF files, 158
P-CAD user-defined attribute translation, 163
PCB 386, 183
  installing a PCB 386 library, 184
PCB 386+
  exporting from Layout, 208
PCB 386+ command (Export), 859
PCB 386+ command (Import), 853
PCB II, 126, 852
PCB II Netlist command (Import), 852
pcb systems home, 73
PDF
  creating from a catalog, 566
PDIF files, 159, 161, 854
pin
  disconnect, 946
  packages, 81
Pin Attachment dialog box, 766
Pin command, 55
pin cost, 977, 1009
pin group, 81
pin swapping, 357
  resistor packages, 436
pin tool
  pop-up commands, 1014
PINGROUP, 505
pins
  adding and deleting, 431
  adding to footprints, 539
  assigning padstacks to, 549
  attaching obstacles to, 551
  checking before component placement, 327
  copying, 936
  deleting, 938
  disconnecting from nets, 432
  matching footprint pin names to schematic pin numbers, 41
  power and ground fanout, 392
  properties, 937
  rotating, 919, 926, 937, 963, 970
  selecting a padstack for Pin 1 in the Pad Array Generator, 541
  swapping, 928
place
  array, 988
  board, 987
  component, 988
  free via matrix, 989
  matrix, 989
  test points, 989
Place Clusters pass, 352, 357
Place command, 337, 352, 417
Place command (Component), 924
place grid, 325
place outline, 84, 684
place outlines, 359, 361
  verifying before component placement, 324
place pass
  Adjust Comps, 353, 356
  Assign Clusters, 352, 356
  editing, 699
  enabling, 355
  information, editing, 353
  Place Clusters, 352, 357
  Proximity Place, 352, 356
  Swap Comps, 353, 357
  Swap Pins, 357
Place Pass command, 74
Place Pass spreadsheet, 91
place pass strategy, 1010
Place Settings command, 977
Place Settings dialog box, 767
placement
  algorithms, 357
  attempts, 358
  circular, 626

clusters, 360, 362
fast reconnect, 357
iterations, 357
preparing the board, 349
push-and-shove, 351
statistics, 376
strategy files, 1010
strategy spreadsheet, 74
using autoplacement, 352
using circular placement, 363, 365
placement strategy
editing, 699
Placement Strategy command, 74, 976
placement strategy files, 132
placing, 898
component clusters, 337
grouping components, 332
placing components, 818
description, 27
in a matrix, 342
in groups, 338
individually, 335
manually, 335
minimizing connections, 339
preparing the board for, 323
securing preplaced components, 329
selecting the next component, 336
spacing violations, 374
plane
manually routing connected nets, 384
settings, 981
split, converting, 220
plane layer, 740
creating split planes, 398
plane layers
verifying net connections, 401
plot/print, 774
plotting, 893
Gerber plot preview, 515
pointer and cursor information, 69
pop-up
end command, 1018
pop-up commands, 1010
component tool, 1012
connection tool, 1015
error tool, 1015
gate tool, 1013
obstacle tool, 1014
pin tool, 1014
text tool, 1015
track segment tool, 1017
track tool, 1016
post process, 727, 885
database spreadsheet, 89
DXF, 215
load, 888
run, 1002
save as, 892
Post Process command, 91
Post Process Settings command, 75, 91, 519, 983
Post Process Settings dialog box, 769
Post Process spreadsheet, 91
post processing, 519
in design flow, 29
preview, 509
pour, 398
power
enabling for routing, 385
routing, 382
on SMT boards, 382
on through-hole boards, 382
verifying connection to plane layer, 401
power and ground fanout, 392
power nets, disabling, 351
POWERPIN, 504
PPF files, 119
preferences
3D effects, 93
activate selection filter, 95
autotool select, 94
copper pour for connectivity, 94
fast fill copper pour, 94
footprint editing, 93
full screen cursor, 92
hollow pads, 93

online DRC, 93
opaque graphics, 93
reconnect mode, 93
saving, 95
setting, 92
tooltips, 94
track width, 95
user, 983
user preferences, 842
preferred thermal reliefs, 461
preplaced components, securing, 329
Preview command, 509
Preview command (View menu), 904
preview, post processing, 509
previewing
Gerber, 904
Print/Plot command (File menu), 893
Print/Plot dialog box, 774
printing, 893
Gerber plot preview, 515
problems
known problems and solutions, 73
product notes, 73
Product Notes command, 1006
properties
component, 929
dimension, 965
drill, 655
drill chart, 654
editing, 901
jumper, 954
layer, 911
nets, 943
test point, 959
via, 957
Properties command, 54, 91, 229, 242, 248–249, 251, 256, 344, 419, 431, 461, 467, 494, 519
Properties command (Aperture), 938
Properties command (Component), 929
Properties command (Dimension), 965
Properties command (Edit menu), 901
Properties command (Group), 920
Properties command (Jumper), 954
Properties command (Layer), 911
Properties command (Net), 943
Properties command (Padstack), 934
Properties command (Pin), 937
Properties command (Test Point), 959
Properties command (Via), 957
property name, 502
Protel 99SE import, 170
Protel 99SE INI file, 170
Protel 99SE Input/Output files, 170
Protel 99SE layer mapping (Import), 171
Protel import, 166
Protel INI file, 167, 201
Protel Input/Output files, 167
Protel Input/Output files (Export), 201
Protel layer mapping (Export), 202
Protel layer mapping (Import), 167
Protel PCB command (Export), 860
Protel PCB command (Import), 854
Protel PCB file, 201
Protel99 SE command (Import), 854
Proximity Place pass, 352, 356
push-and-shove placement, 351

Q

QFP/Chip Carrier tab (Pad Array Generator), 780
quad flat pack pad array settings, 780
queries, 719
Query Window command, 55, 57, 905
querying, 905
flagged errors, 474
spreadsheet information, 58
Queue For Placement command, 335
Queue for Placement command (Component), 924
Quick Place command, 363
quit Layout, 895

R

radius, text, 267

ratsnest
description of, 378
making invisible, 428
manual routing, 394
zero-length connection, 378
reconnect mode, 93, 358
Reconnection Type dialog box, 783
reconnection types
high speed, 250
horizontal, 250
no dynamic reconnect, 251, 382
none, 249
standard orthogonal, 250
vertical, 250
RECONNTYPE, 504
Redraw command (View menu), 905
redraw screen, 905
reference designator, 344, 720
refresh, 905
all, 985
Refresh command, 56
relative dimension, 505
Release 9 format
updating boards, 217
release notes, 1006
Reload App Settings command (Tools menu), 881
Remove Tack Point command, 420
removing errors, 1000
Rename Components command, 494, 1002
Rename Direction dialog box, 785
renaming
components, 982
renaming components, 493
repaint, 905
Replace dialog box, 786
report
post process, 89
report definition, 520
reports
creating, 1003
generating, 455, 727
reports, creating, 520
repour, 985
Repour command, 986
Reset All command (Window menu), 1005
resistor package
pin swapping, 437
resistor package pin swap option #1, 436
resistor package pin swap option #2, 437
resistor packages, 436
swapping, 436
resizing the DRC box, 387
Resume Routing command, 996
reusing .MAX files, 887
reusing MAX files, 348
rotate
board, 915
track, 948
Rotate command, 258, 341, 919, 926, 963, 970
Rotate command (Block), 915
Rotate command (Pin), 937
Rotate command (Track), 948
rotating
pins, 937
rotation, 345
components, 341
text, 267
rotation settings, 974
round board, placing components on, 363, 365
route
routing to an off-grid pad, 421
route layer, 977, 1009
Route Layer command, 977, 1009
Route Layer spreadsheet, 75
route layer strategy, 379
route pass
editing, 703
Route Pass command, 980, 1008
Route Pass spreadsheet, 75
Route Settings command, 242, 407, 409, 980
Route Settings dialog box, 789
Route Spacing spreadsheet, 75

route spacing spreadsheet, 1010
Route Strategies command, 74
route strategy, 703
    edit spacing, 709
Route Sweep command, 978
Route Sweep spreadsheet, 75
route sweep strategy, 379
route-enabled components, 347
routing, 949, 955, 994–995
    45 corners, 402
    90 corners, 402
    adding a via matrix, 417
    adding tacks, 420
    adding vias, 417
    any angle corner, 402
    auto path route, 406
    changing net colors, 412
    changing the layer of a routed net, 396
    changing track width, 415
    checking
        board outline, 378
        routing grid, 378
        via definitions, 378
        via grid, 378
    copying tracks, 412
    creating a track with curved corners, 396
    creating duplicate connections, 410
    curve corners, 402
    density graph, 70
    description, 27
    editing free vias, 419
    editing strategy files, 379
    editing vias, 418
    enabling a layer for routing, 977, 1009
    enabling layers for routing, 248
    Exchange Ends command, 421
    fanout, 388
    forcing a net width on a layer, 416
    global spacing, 976
    interactive, 406
    jumpers, 741
    layer strategy, 675
    locking tracks, 421
    making vias available for, 242
    manual, 394
    manual route strategy, 977
    minimizing connections, 411
    net spacing by layer, 757
    net widths by layer, 758
    obstacles, 84, 971
    power and ground nets, 382
        on SMT boards, 382
        on through-hole boards, 382
    removing tacks, 420
    resume, 996
    route settings, 789
    setting net spacing by layer, 251
    setting net width, 248–249
    shove track, 406
    shoving routes interactively, 407, 409
    snap to grid, 1017
    snap to grid routing, 402
    spacing violations, 434–435
    strategy, 753
    strategy spreadsheets, 74
    sweep edit, 829
    teeing into or out of an existing track, 425
    T-routing, 402
    unlocking tracks, 421
    Unroute Center Partial command, 413
    Unroute command, 413
    Unroute Net command, 413
    Unroute Partial Track command, 413
    Unroute Segment command, 413
    Unroute Unlocked Track command, 413
    use all via types, 402
routing attempts, 703
routing hints, 428
routing modes
    add/edit route mode, 403
    auto path route mode, 409
    edit segment mode, 404
    interactive, 439
    shove track mode, 407
routing nets

using SPECCTRA, 448
routing strategy spreadsheets, 74
rules
check design rules, 1000
colors, 607
Run Post Processor command, 519, 1002

S

save, 622, 889, 975
Save As
Layout v7.0 board, 890
strategy file, 891
save as, 889
aperture list, 890
no extension records, 890
Save As Color Setup command, 891
Save As command, 46, 227–228
Save As command (File menu), 889
Save As Post Processing Setup command, 892
Save As Strategy command, 891
Save As v7.0 command, 890
Save command, 46, 54
Save command (File menu), 889
Save Footprint As dialog box, 812
Save padstack - Select library dialog box, 813
Save to Library command, 934
saving
board files, 46
preferences, 95
schematic
forward annotation, 496
package editing, 760
schematic attributes, 502
schematic interface, 495
schematic netlist
comparing to a layout board, 502
screen
half screen window, 1004
searching, 897
reference designator or coordinate, 720
seed
copper pour seed, 937
seed point, designating for copper pour, 464
seeding, copper pour, 466
segment
editing, 406
Segment command, 259
Segment command (Track), 1018
segments, moving, 259
select, 899
components, 898
Select All command (Block), 914
Select Any command, 247, 338, 362
Select Any command (Edit menu), 898
select entire track, 911
Select Filtered command (Component), 924
Select Footprint dialog box, 816
select from invisible layers, 911
Select Layer command, 257, 911
Select Layer dialog box, 818
select locked and fixed, 911
select new footprint, 720
Select Next command, 335
Select Next command (Edit menu), 899
Select Next dialog box, 818
Select Padstack dialog box (Pad Array Generator), 820
Select Padstacks dialog box, 819
Select Tool command (Block), 914
Select Tool command (Component), 923
Select Tool command (Connection), 944
Select Tool command (Dimension), 964
Select Tool command (Error), 972
Select Tool command (Gate), 930
Select Tool command (Jumper), 952
Select Tool command (Matrix), 921
Select Tool command (Measurement), 966
Select Tool command (Obstacle), 968
Select Tool command (Pin), 936
Select Tool command (Test Point), 958
Select Tool command (Text), 961
Select Tool command (Track Segment), 950

Select Tool command (Track), 947
Select Tool command (Via), 955
selecting, 68
  area, 61
  deselecting, 61
  multiple objects, 60–61
  nets, 756
  obstacle segments, 256
  obstacles, 255, 759
  one object, 60
  text, 839
  using autotool select, 59–60, 94
  using selection filter, 59
  using tools, 59–61
selecting items, 719
selection criteria
  component, 637
  test point, 836
selection filter, 95
  activate, 64
  example, 65
  shortcut keys, 64
  using, 64
session log, 52
session window, 49
settings, 974
  backup, 622
  editing application settings, 880
  free vias, 982
  Gerber, 983
  jumpers, 981
  post process, 983
  reloading application settings, 881
  route, 980
  test points, 982
  user preferences, 983
shield
  copper pour, 469
shortcut keys, 109
  auto menu, 114
  edit menu, 111
  file menu, 111
  global, 109
  layer number, 116
  options menu, 114
  tool menu, 112
  view menu, 111
  window and spreadsheet, 115
  window menu, 115
shortcuts, 109, 111–112, 114–116
Shove command, 351, 359
Shove command (Component), 927
shove technology, defined, 438
shove track mode, 394, 407
  activating, 56
  high power, 407
  low power, 407
  medium power, 407
shoving components, 359
shoving routes interactively, 407, 409
signal name
  nets, 83
SmartRoute, 868
SmartRoute command, 868
SMD pads, 712, 981
Snap To Grid command (Track), 1017
solder mask, 827
solder paste, 827
solutions
  known problems and solutions, 73
spacing
  global, 976
  net, 757
  nets, 251
  route strategy, 709
spacing violations
  placement, 374
  routing, 434
SPACINGBYLAYER, 504
SPECCTRA, 751, 825
  autoroute board, 450
  autorouting, 441
  Component Placement report, 823
  Component Property report, 824
  Conflicts report, 823
  Corner report, 823

documentation, 442
exporting Layout to SPECCTRA, 187
fanout, 447
importing into Layout, 181
Keepout report, 823
launch from Layout, 444
miter wire corners, 449
Network report, 824
Padstacks report, 824
Pins without fanout vias report, 823
reports, 455
Route Status report, 823
routing nets, 448
Unconnected Nets report, 823
using customized .DO file, 451
SPECCTRA to Layout command (Import), 855
SPECCTRA to Layout dialog box, 825
split plane
converting pre-Layout v7.10 split planes, 220
split planes, 398
creating, 398
splitting nets, 431
spreadsheet, 977, 1009
apertures, 87
clear selections, 900
color, 88
colors, 974
components, 82
drills database spreadsheet, 86
error markers, 85
footprints, 80
layers, 77
nets, 83
obstacles, 84
packages, 81
padstacks, 78
place pass, 976
post process, 89
route pass, 980, 1008
route spacing, 1010
route sweep, 978
statistics, 76
strategy, 1008
text, 84
spreadsheets, 906
Color, 96
descriptions, 74
displaying, 74
editing information, 75
Padstacks, 558
Place Pass, 91
placement strategy, 74
Post Process, 91
querying information, 58
Route Layer, 75
Route Pass, 75
Route Spacing, 75
Route Sweep, 75
routing strategies, 74
stackup editor, 826
standard orthogonal reconnection type, 250
starting, library manager, 530
startup, 49, 122
statistics
placement, 376
refresh, 986
routing, 435
statistics database spreadsheet, 76
status bar, 57
viewing, 864
Status Bar command, 864
stop a command, 900
stop autorouting, 995
strategy
placement, 976, 1010
renaming components, 982
route spacing, 1010
route sweep, 978
routing, 753
spreadsheet, 1008
Strategy command, 74
strategy file
editing, 379
save as, 891

strategy files, 132, 885
description, 132
list of placement strategy files, 132
list of routing strategy files, 134
loading, 331, 380
placement, 331
PLSTD.SF, 331
string, text, 265
Swap command, 340, 928
Swap Comps pass, 353, 357
Swap Pins pass, 357
swapping
components, 353, 357
gates, 357
pins, 357, 436
resistor packages, 436
sweep
route, 978
Sweep Edit dialog box, 829
sweep technology, defined, 438
symbol data format, 151
System Settings command, 233, 235, 258, 326, 974
System Settings dialog box, 834
SYSTEM.PRT, 119, 122
SYSTEM.PRT files, 122

T

Tack command, 420
Tack command (Connection), 945
tacking connections, 945
Tango import, 177
Tango INI file, 178, 204
Tango Input/Output files (Import), 178
Tango Input/Output files(Export), 204
Tango layer mapping (Export), 205
Tango layer mapping (Import), 178
Tango PCB command (Export), 861
Tango PCB command (Import), 854
targets
creating alignment targets, 261
technology file
loading, 886
technology templates, 32, 127, 225
description, 127
list of, 127
teeing into or out of an existing track, 425
template file
loading, 886
template files, 885
templates
board, 32, 127, 225
creating custom, 226
technology, 32, 127, 225
test point
add, 959
add test point dialog box, 617
delete, 959
delete all, 960
new, 958
properties, 959
select tool, 958
selection criteria, 836
settings, 835
Test Point Selection Criteria dialog box, 836
Test Point Settings command, 982
Test Point Settings dialog box, 835
test points
editing, 711
generating, 432
Test Points command (Place), 989
TESTPOINT, 504
text
character aspect, 267
character rotation, 267
component value, 266
copy, 961
creating, 263
custom properties, 266
delete, 963
deleting, 268
footprint name, 266
free, 265
labeling footprints, 555
location, 267
mirroring, 258, 918, 925, 961–962, 969

moving, 268
new, 961
package name, 266
radius, 267
rotating, 919, 926, 963, 970
rotation, 267
select tool, 961
string, 265
width of line, 267
Text command, 55
text database spreadsheet, 84
Text Edit dialog box, 837
text editor, 895
Text Editor command, 52, 895
Text Selection Criteria dialog box, 839
text tool
pop-up commands, 1015
Thermal Relief Settings command, 458, 981
Thermal Relief Settings dialog box, 840
thermal reliefs, 382, 392, 712, 740
annular over drill, 458
creation rules, 460
defining, 458
forced, 461
isolation width, 458
preferred, 461
previewing, 459
spoke width, 458
using padstacks, 462
THERMALLAYERS, 504
thrucodes, 80, 559, 690, 695
Tile command (Window menu), 1004
Toggle Copper Pour Seed command, 464, 937
tool menu shortcut keys, 112
toolbar
add/edit route, 56
auto path route, 56
color, 55
component, 55
connection, 55
delete, 54
design rule check, 56
edit, 54
edit segment, 56
error, 55
layer drop-down list, 72
library manager, 54
obstacle, 55
online DRC, 56
open, 54
pin, 55
postage stamp view, 73
query, 55
reconnect, 56
refresh all, 56
save, 54
selection filter, 57
shove track, 56
spreadsheet, 54
text, 55
tooltips, 54
viewing, 864
viewing object coordinates, 72
viewing place grid setting, 72
zoom all, 55
zoom in, 54
zoom out, 55
Toolbar command (View menu), 864
tooltips, 54, 94
disabling, 54
track
45 corners, 1017
90 corners, 1017
any angle corners, 1018
change width, 942
curve corners, 1018
finish, 971
lock, 949
mirror, 948
new, 1018
rotate, 948
segment, 1018
select tool, 947
snap to grid, 1017

teeing into or out of, 425
unlock, 949
unroute, 950
unroute segment, 1018
track segment
change width, 951
delete, 951
exchange ends, 950
select tool, 950
track segment tool
pop-up commands, 1017
track to pad spacing, 709
track to track spacing, 709
track to via spacing, 709
track tool
pop-up commands, 1016
track width, 396–397
minimum, 95
Track Width dialog box, 841
tracks
curved corners, 396
force minimum and maximum track width, 943
force width by layer, 942
locking, 920, 928
manual routing, 394
rotating, 919, 926, 963, 970
tracks, copying, 412
tranlating
CadStar files, 173
translating, 856
CadStar PCB files, 191–192, 853
DXF files, 209, 211, 855
Futurenet Netlists, 852
GenCAD files, 855
GenCAD to Layout, 182, 724
IDF files, 855
IDF to Layout, 165, 738
Layout to GenCAD, 188, 744
Layout to GenCAM, 189
exporting
Layout to GenCAM, 745
Layout to Hyperlynx, 207, 743
Layout to IDF, 186, 747
Layout to IPC-356, 190, 746
Layout to ODB++, 194, 749
Layout to PCB 386+, 208
Layout to SPECCTRA, 187, 751
MAX ASCII Netlists, 852
MIN interchange, 850
OrCAD MIN ASCII file, 857
PADS files, 154
PADS PCB files, 154–155, 195–196, 853, 859
P-CAD PCB files, 158–159, 161, 198–199, 854, 860
P-CAD user-defined attributes, 163
PCB 386 files, 183
PCB 386+ commands, 853
PCB II Netlists, 852
PCB386 libraries, 184
Protel 99SE files, 170
Protel 99SE PCB files, 170–171, 854
Protel files, 166
Protel PCB files, 167, 854
SPECCTRA files, 855
SPECCTRA to Layout, 181, 825
Tango files, 177
Tango PCB files, 178, 854
to CadStar PCB files, 859
to DXF files, 861
to GenCAD files, 862
to GenCAM format, 863
to Hyperlynz files, 862
to IDF files, 862
to IPC-356 files, 862
to MAX ASCII netlists, 858
to ODB++ format, 863
to PCB 386+ files, 859
to Protel PCB files, 201–202, 860
to SPECCTRA format, 863
to Tango PCB files, 204, 861
mapping
Tango layers, 205
translating a DXF file, 209
translating CadStar PCB files, 174

triangles, 384
indicate unrouted zero-length connections, 955
T-routing, 250, 402
tutorial, 73, 1007
tutorial, Layout online, 73
type of via
changing, 956

U

undo, 896
Undo command (Edit menu), 896
units of measurement, 231
Unlock command, 421
Unlock command (Net), 941
Unlock command (Track), 949
unplace
board, 990
component, 990
unroute
board, 999
component, 999
DRC/Route Box, 999
nets, 999
Unroute Center Partial command, 413
Unroute command, 413
Unroute command (Layer), 911
Unroute command (Net), 940
Unroute command (Track), 950
Unroute Net command, 413
Unroute Partial Track command, 413
Unroute Partial Track command (Net), 940
Unroute Segment command, 413
Unroute Segment command (Track), 1018
Unroute Unlocked Track command, 413
unused objects, 632
UPDATE90.EXE, 217
updating boards and libraries to Release 9 format, 217
User Preferences command, 92, 358–359, 983
User Preferences dialog box, 842
USER.PRT files, 123
user-defined attribute translation from P-CAD, 163
using copper pour as a shield, 469

V

value, component, 345
vertical reconnection type, 250
via
adding, 955
adding free vias, 956
assign via dialog box, 619
BGA fanout, 390
change via type, 956
delete, 957
modify padstacks, 957
new, 955
properties, 957
select tool, 955
via cost, 703
via fanout, 712
via keepout, 84
via reduce, 703
Via Selection dialog box, 847
via to pad spacing, 709
via to via spacing, 709
via-in-pad technology (fanning out BGAs), 390
VIAPERNET, 504
vias, 242, 244, 389, 417–418
assign via per net, 941
assigning to nets, 243–244
checking definitions, 378
defining an unused via, 242
editing free vias, 668
free, 242, 244, 389, 417, 419
generating test points, 432
test points, 711
view
ratsnest, 428
view menu shortcut keys, 111
viewing
fit all in window, 906
print/plot, 774

viewing a Layout netlist file, 139
viewing drill holes, 245
visibility
  assigning a color to a layer, 98
  assigning a color to an object, 97
  objects and layers, 99
visibility and color, 361
Visible Invisible command (View menu), 909
visible, making layers, 99
VisibleInvisible command, 361

W

warning
  library conversion, 752
web resources
  customer support, 73
  pcb systems home, 73
Web Resources command, 1006
weighting nets, 326
WIDTH, 504
width
  force minimum and maximum track width, 943
  net, 758
  nets, 248–249
  text line, 267
  track, 396–397, 841
  track width by layer, 942
WIDTHBYLAYER, 505
window
  density graph, 70
window and spreadsheet shortcuts, 115
window menu shortcut keys, 115
windows, 1004
  1, 2, 3, 4 commands, 1005
  arrange icons, 1004
  cascade, 1004
  design, 50
  footprint editor, 51
  library manager, 51
  reset all, 1005
  session log, 52
  spreadsheet, 74
  tiling, 1004
workspace settings, 974
Workspace Settings dialog box, 847

Y

yellow triangle, 955
yellow triangles, 384
yellow triangles in ratsnest, 378

Z

z order, copper pour, 466
zero-length connection, 378
zone, 398, 469, 740
zoom, 105, 906–909
Zoom All (Fit) command (View menu), 906
Zoom All command, 55, 227
Zoom Center command (View menu), 907
Zoom DRC/Route Box command, 387
Zoom DRC/Route Box command (View menu), 909
Zoom In command, 54
Zoom In command (View menu), 907
Zoom Out command, 55, 254
Zoom Out command (View menu), 908
Zoom Previous command (View menu), 908